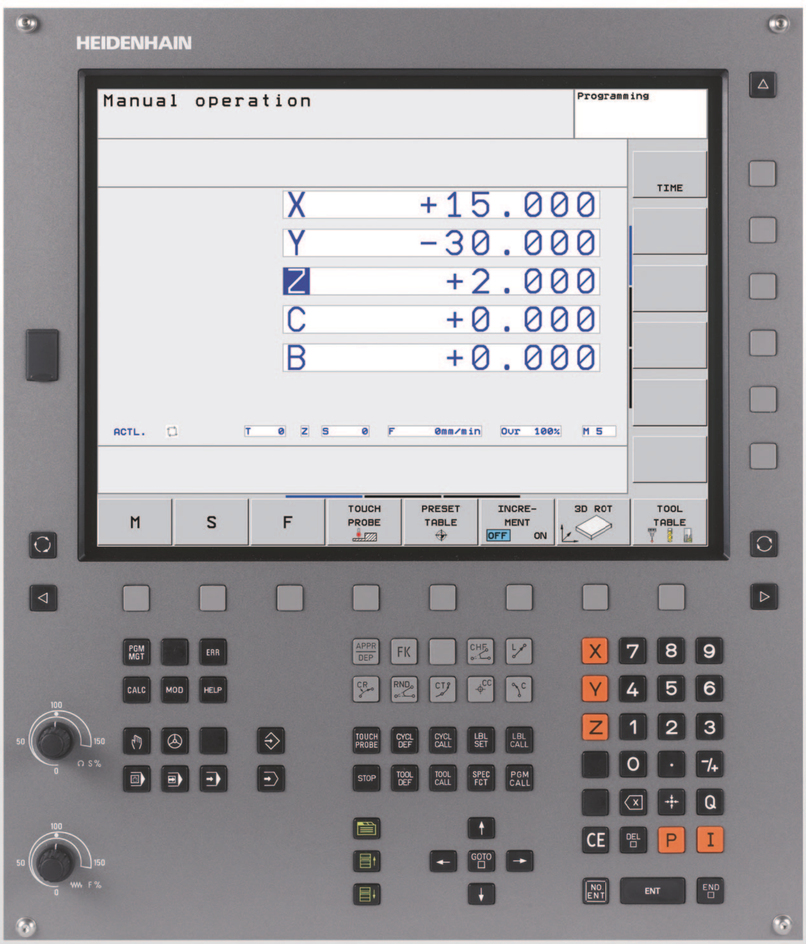
HEIDENHAIN



User's Manual
ISO Programming

TNC 320

NC Software
340 551-05
340 554-05

English (en)
11/2011

Controls of the TNC

Keys on visual display unit

Key	Function
	Split screen layout
	Toggle the display between machining and programming modes
	Soft keys for selecting functions on screen
	Switching the soft-key rows

Machine operating modes

Key	Function
	Manual Operation
	Electronic Handwheel
	Positioning with Manual Data Input
	Program Run, Single Block
	Program Run, Full Sequence

Programming modes

Key	Function
	Programming and Editing
	Test Run

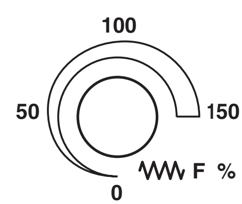
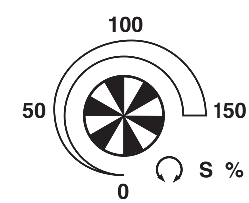
Program/file management, TNC functions

Key	Function
PGM MGT	Select or delete programs and files, external data transfer
PGM CALL	Define program call, select datum and point tables
MOD	Select MOD functions
HELP	Display help text for NC error messages, call TNCguide
ERR	Display all current error messages
CALC	Show calculator

Navigation keys

Key	Function
	Move highlight
GOTO	Go directly to blocks, cycles and parameter functions

Potentiometer for feed rate and spindle speed

Feed rate	Spindle speed
	

Cycles, subprograms and program section repeats

Key	Function
TOUCH PROBE	Define touch probe cycles
CYCL DEF / CYCL CALL	Define and call cycles
LBL SET / LBL CALL	Enter and call labels for subprogramming and program section repeats
STOP	Program stop in a program

Tool functions

Key	Function
TOOL DEF	Define tool data in the program
TOOL CALL	Call tool data

Programming path movements

Key	Function
APPR DEP	Approach/depart contour
FK	FK free contour programming
L	Straight line
CC	Circle center/pole for polar coordinates
C	Circle with center
CR	Circle with radius
CT	Circular arc with tangential connection
CHF RND	Chamfering/Corner rounding

Special functions

Key	Function
SPEC FCT	Show special functions
	Select the next tab in forms
	Up/down one dialog box or button

Coordinate axes and numbers: Entering and editing

Key	Function
X . . . V	Select coordinate axes or enter them into the program
0 . . . 9	Numbers
. −/+	Decimal point / Reverse algebraic sign
P I	Polar coordinate input / Incremental values
Q	Q-parameter programming / Q parameter status
	Save actual position or values from calculator
NO ENT	Skip dialog questions, delete words
ENT	Confirm entry and resume dialog
END	Conclude block and exit entry
CE	Clear numerical entry or TNC error message
DEL	Abort dialog, delete program section

About this Manual

The symbols used in this manual are described below.

This symbol indicates that important notes about the function described must be adhered to.

This symbol indicates that there is one or more of the following risks when using the described function:

- Danger to workpiece
- Danger to fixtures
- Danger to tool
- Danger to machine
- Danger to operator

This symbol indicates that the described function must be adapted by the machine tool builder. The function described may therefore vary depending on the machine.

This symbol indicates that you can find detailed information about a function in another manual.

Would you like any changes, or have you found any errors?

We are continuously striving to improve our documentation for you. Please help us by sending your requests to the following e-mail address: **tnc-userdoc@heidenhain.de**.

TNC Model, Software and Features

This manual describes functions and features provided by TNCs as of the following NC software numbers.

TNC model	NC software number
TNC 320	340 551-05
TNC 320 Programming Station	340 554-05

The machine tool builder adapts the usable features of the TNC to his machine by setting machine parameters. Some of the functions described in this manual may therefore not be among the features provided by the TNC on your machine tool.

TNC functions that may not be available on your machine include:

- Tool measurement with the TT

Please contact your machine tool builder to become familiar with the features of your machine.

Many machine manufacturers, as well as HEIDENHAIN, offer programming courses for the TNCs. We recommend these courses as an effective way of improving your programming skill and sharing information and ideas with other TNC users.

User's Manual for Cycle Programming:

All of the cycle functions (touch probe cycles and fixed cycles) are described in a separate manual. Please contact HEIDENHAIN if you need a copy of this User's Manual. ID: 679 220-xx

Software options

The TNC 320 features various software options that can be enabled by your machine tool builder. Each option is to be enabled separately and contains the following respective functions:

Hardware options
Additional axis for 4 axes and open-loop spindle
Additional axis for 5 axes and open-loop spindle

Software option 1 (option number #08)
Cylinder surface interpolation (Cycles 27, 28 and 29)
Feed rate in mm/min for rotary axes: **M116**
Tilting the machining plane (plane functions, Cycle 19 and 3D-ROT soft key in the Manual Operation mode)
Circle in 3 axes with tilted working plane

Feature content level (upgrade functions)

Along with software options, significant further improvements of the TNC software are managed via the Feature Content Level **(FCL)** upgrade functions. Functions subject to the FCL are not available simply by updating the software on your TNC.

All upgrade functions are available to you without surcharge when you receive a new machine.

Upgrade functions are identified in the manual with **FCL n,** where **n** indicates the sequential number of the feature content level.

You can purchase a code number in order to permanently enable the FCL functions. For more information, contact your machine tool builder or HEIDENHAIN.

Intended place of operation

The TNC complies with the limits for a Class A device in accordance with the specifications in EN 55022, and is intended for use primarily in industrially-zoned areas.

Legal information

This product uses open source software. Further information is available on the control under

- Programming and Editing operating mode
- MOD function
- LICENSE INFO soft key

New Functions of Software 340 55x-04

- The **PLANE** function for flexible definition of a tilted working plane was introduced (see "The PLANE Function: Tilting the Working Plane (Software Option 1)" on page 299).
- The context-sensitive help system TNCguide was introduced (see "Calling the TNCguide" on page 124).
- The conversational languages Estonian, Korean, Latvian, Norwegian, Romanian, Slovak and Turkish were introduced (see "Parameter list" on page 416).
- Individual characters can now be deleted by using the backspace key (see "Coordinate axes and numbers: Entering and editing" on page 3).
- The **PATTERN DEF** function for defining point patterns was introduced (see User's Manual for Cycles).
- The **SEL PATTERN** function makes it possible to select point tables (see User's Manual for Cycles).
- With the **CYCL CALL PAT** function, cycles can now be run in connection with point tables (see User's Manual for Cycles).
- The **DECLARE CONTOUR** function can now also define the depth of the contour (see User's Manual for Cycles).
- New Cycle 241 for Single-Lip Deep-Hole Drilling was introduced (see User's Manual for Cycles).
- The new fixed cycles 251 to 257 were introduced for milling pockets, studs and slots (see User's Manual for Cycles).
- Touch probe cycle 416 (Datum on Circle Center) was expanded by parameter Q320 (safety clearance) (see User's Manual for Cycles).
- Touch probe cycles 412, 413, 421 and 422: Additional parameter Q365 (type of traverse) (see User's Manual for Cycles).
- Touch probe cycle 425 (Measure Slot) was expanded by parameters Q301 (Move to clearance height) and Q320 (setup clearance) (see User's Manual for Cycles).
- Touch probe cycles 408 to 419: The TNC now also writes to line 0 of the preset table when the display value is set (see User's Manual for Cycles).
- Datum tables can now also be selected in the machine operating modes Program Run, Full Sequence and Program Run, Single Block (**STATUS M**).
- The definition of feed rates in fixed cycles can now also include **FU** and **FZ** values (see User's Manual for Cycles).

Changed Functions of Software 340 55x-04

- In Cycle 22 you can now define a tool name also for the coarse roughing tool (see User's Manual Cycles).
- The additional status display has been revised. The following improvements have been made (see "Additional status displays" on page 63):
  - A new overview page with the most important status displays was introduced.
  - The tolerance values set in Cycle 32 are displayed.
- The pocket-, stud- and slot-milling cycles 210 to 214 were removed from the standard soft-key row (CYCL DEF > POCKETS/STUDS/SLOTS). For reasons of compatibility, the cycles will still be available, and can be selected via the GOTO key.
- With Cycle 25 Contour Train, closed contours can now also be programmed.
- Tool changes are now also possible during mid-program startup.
- Language-sensitive texts can now be output with FN16 F-Print.
- The soft-key structure of the SPEC FCT function was changed and adapted to the iTNC 530.

New Functions of Software 340 55x-05

- The **M101** function was introduced (see "Automatic tool change if the tool life expires: M101" on page 146).
- Tool tables of the iTNC 530 can now be imported in the TNC 320 and converted to a valid format (see "Importing tool tables" on page 140).
- The **CYCL CALL POS** function was introduced (see User's Manual for Cycles).
- Local and nonvolatile **QL** and **QR** Q parameters were introduced (see "Principle and Overview" on page 202).
- You can now perform a tool usage test before starting the program (see "Tool usage test" on page 148).
- The M138 "Selection of swivel axes" function was introduced (see "Selecting tilting axes: M138" on page 323).
- File functions were introduced (see "File Functions" on page HIDDEN).
- The "Define coordinate transformations" function was introduced (see "Defining Coordinate Transformations" on page HIDDEN).

Changed Functions of Software 340 55x-05

- The status display for Q parameters was improved (see "Checking and Changing Q Parameters" on page 211).
- The LAST_USE column was added to the tool table (see "Tool table: Standard tool data" on page 134).
- The graphic simulation was enhanced and adapted to the iTNC 530 (see "Graphics" on page 368).
- Touch probe cycles can now also be used in the tilted working plane (see User's Manual for Cycles).

Contents

1 First Steps with the TNC 320 33

2 Introduction 53

12 Manual Operation and Setup 325

1

First Steps with the TNC 320

1.1 Overview

This chapter is intended to help TNC beginners quickly learn to handle the most important procedures. For more information on a respective topic, see the section referred to in the text.

The following topics are included in this chapter:

- Machine Switch-On
- Programming the First Part
- Graphically Testing the First Program
- Tool Setup
- Workpiece Setup
- Running the First Program

1.2 Machine Switch-On

Acknowledge the power interruption and move to the reference points

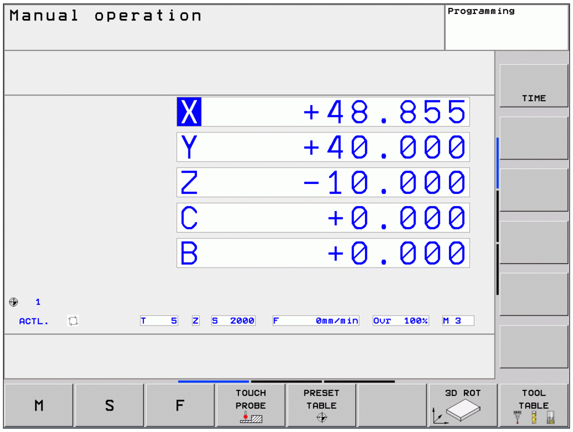
> Switch-on and crossing the reference points can vary depending on the machine tool. Your machine manual provides more detailed information.

▶ Switch on the power supply for control and machine. The TNC starts the operating system. This process may take several minutes. Then the TNC will display the message "Power interruption."

CE ▶ Press the CE key: The TNC converts the PLC program

I ▶ Switch on the control voltage: The TNC checks operation of the emergency stop circuit and goes into the reference run mode

I ▶ Cross the reference points manually in the displayed sequence: For each axis press the machine START button. If you have absolute linear and angle encoders on your machine there is no need for a reference run



The TNC is now ready for operation in the **Manual Operation** mode.

Further information on this topic

- Traversing the reference marks: See "Switch-on" on page 326
- Operating modes: See "Programming and Editing" on page 59

1.3 Programming the First Part

Select the correct operating mode

You can write programs only in the Programming and Editing mode:

▶ Press the operating modes key: The TNC goes into the **`Programming and Editing`** mode

Further information on this topic

- Operating modes: See "Programming and Editing" on page 59

The most important TNC keys

Functions for conversational guidance	Key
Confirm entry and activate the next dialog prompt	ENT
Ignore the dialog question	NO ENT
End the dialog immediately	END
Abort dialog, discard entries	DEL
Soft keys on the screen with which you select functions appropriate to the active state	

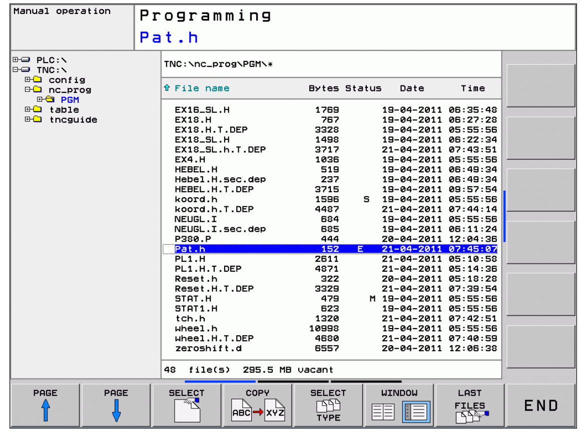
Further information on this topic

- Writing and editing programs: See "Editing a program" on page 84
- Overview of keys: See "Controls of the TNC" on page 2

Create a new program/file management

PGM MGT

- ▶ Press the PGM MGT key: the TNC displays the file management. The file management of the TNC is arranged much like the file management on a PC with the Windows Explorer. The file management enables you to manipulate data on the TNC hard disk
- ▶ Use the arrow keys to select the folder in which you want to open the new file
- ▶ Enter a file name with the extension **.I**: The TNC then automatically opens a program and asks for the unit of measure for the new program
- ▶ To select the unit of measure, press the MM or INCH soft key: The TNC automatically starts the workpiece blank definition (see "Define a workpiece blank" on page 38)



The TNC automatically generates the first and last blocks of the program. Afterwards you can no longer change these blocks.

Further information on this topic

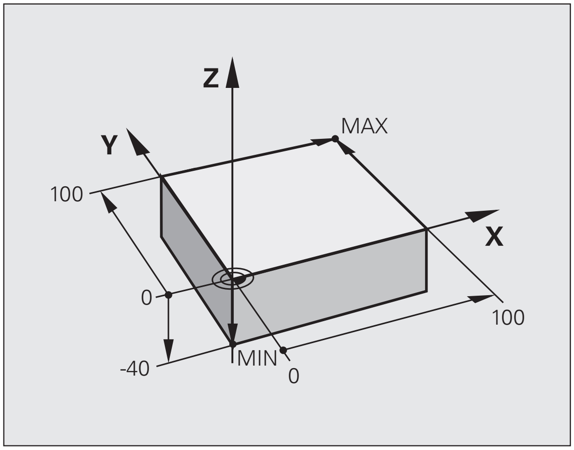
- File management: See "Working with the File Manager" on page 92
- Creating a new program: See "Creating and Writing Programs" on page 79

Define a workpiece blank

Immediately after you have created a new program, the TNC starts the dialog for entering the workpiece blank definition. Always define the workpiece blank as a cuboid by entering the MIN and MAX points, each with reference to the selected reference point.

After you have created a new program, the TNC automatically initiates the workpiece blank definition and asks for the required data:

- **`Spindle axis Z – Plane XY`**: Enter the active spindle axis. G17 is saved as default setting. Accept with the ENT key
- **`Workpiece blank def.: Minimum X`**: Enter the smallest X coordinate of the workpiece blank with respect to the reference point, e.g. 0. Confirm with the ENT key
- **`Workpiece blank def.: Minimum Y`**: Enter the smallest Y coordinate of the workpiece blank with respect to the reference point, e.g. 0. Confirm with the ENT key
- **`Workpiece blank def.: Minimum Z`**: Enter the smallest Z coordinate of the workpiece blank with respect to the reference point, e.g. –40. Confirm with the ENT key
- **`Workpiece blank def.: Maximum X`**: Enter the largest X coordinate of the workpiece blank with respect to the reference point, e.g. 100. Confirm with the ENT key
- **`Workpiece blank def.: Maximum Y`**: Enter the largest Y coordinate of the workpiece blank with respect to the reference point, e.g. 100. Confirm with the ENT key
- **`Workpiece blank def.: Maximum Z`**: Enter the largest Z coordinate of the workpiece blank with respect to the reference point, e.g. 0. Confirm with the ENT key. The TNC concludes the dialog



Example NC blocks

```
%NEW G71 *
N10 G30 G17 X+0 Y+0 Z-40 *
N20 G31 X+100 Y+100 Z+0 *
N99999999 %NEW G71 *
```

Further information on this topic

- Defining the workpiece blank: (see page 80)

Program layout

NC programs should be arranged consistently in a similar manner. This makes it easier to find your place, accelerates programming and reduces errors.

Recommended program layout for simple, conventional contour machining

1. Call tool, define tool axis
2. Retract the tool
3. Pre-position the tool in the working plane near the contour starting point
4. In the tool axis, position the tool above the workpiece, or pre-position immediately to workpiece depth. If required, switch on the spindle/coolant
5. Move to the contour
6. Machine the contour
7. Leave the contour
8. Retract the tool, end the program

Further information on this topic:

- Contour programming: See "Tool Movements" on page 156

Example: Layout of contour machining programs

```
%BSPCONT G71 *
N10 G30 G71 X... Y... Z... *
N20 G31 X... Y... Z... *
N30 T5 G17 S5000 *
N40 G00 G40 G90 Z+250 *
N50 X... Y... *
N60 G01 Z+10 F3000 M13 *
N70 X... Y... RL F500 *
...
N160 G40 ... X... Y... F3000 M9 *
N170 G00 Z+250 M2 *
N99999999 BSPCONT G71 *
```

Recommended program layout for simple cycle programs

1. Call tool, define tool axis
2. Retract the tool
3. Define the fixed cycle
4. Move to the machining position
5. Call the cycle, switch on the spindle/coolant
6. Retract the tool, end the program

Further information on this topic:

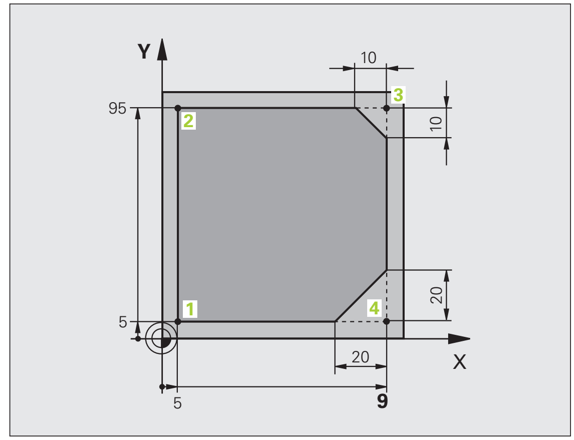
- Cycle programming: See User’s Manual for Cycles

Example: Program layout for cycle programming

```
%BSBCYC G71 *
N10 G30 G71 X... Y... Z... *
N20 G31 X... Y... Z... *
N30 T5 G17 S5000 *
N40 G00 G40 G90 Z+250 *
N50 G200... *
N60 X... Y... *
N70 G79 M13 *
N80 G00 Z+250 M2 *
N99999999 BSBCYC G71 *
```

Program a simple contour

The contour shown to the right is to be milled once to a depth of 5 mm. You have already defined the workpiece blank. After you have initiated a dialog through a function key, enter all the data requested by the TNC in the screen header.



TOOL CALL

- Call the tool: Enter the tool data. Confirm each of your entries with the ENT key. Do not forget the tool axis
- Press the L key to open a program block for a linear movement
- Press the left arrow key to switch to the input range for G codes

G00

- Press the G0 soft key if you want to enter a rapid traverse motion
- Retract the tool: Press the orange axis key Z in order to get clear in the tool axis, and enter the value for the position to be approached, e.g. 250. Confirm with the ENT key
- Confirm **Radius comp.: RL/RR/no comp?** by pressing the ENT key: Do not activate the radius compensation
- Confirm the **Miscellaneous function M?** with the END key: The TNC saves the entered positioning block
- Press the L key to open a program block for a linear movement
- Press the left arrow key to switch to the input range for G codes

G00

- Press the G0 soft key if you want to enter a rapid traverse motion
- Preposition the tool in the working plane: Press the orange X axis key and enter the value for the position to be approached, e.g. –20
- Press the orange Y axis key and enter the value for the position to be approached, e.g. –20. Confirm with the ENT key
- Confirm **Radius comp.: RL/RR/no comp?** by pressing the ENT key: Do not activate the radius compensation
- Confirm the **Miscellaneous function M?** with the END key: The TNC saves the entered positioning block

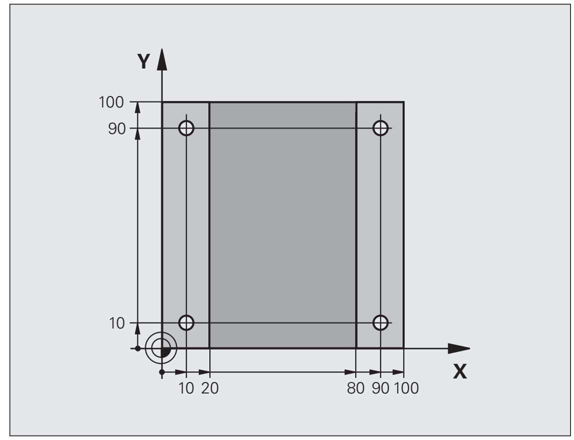
- Move the tool to workpiece depth: Press the orange axis key and enter the value for the position to be approached, e.g. –5. Confirm with the ENT key
- Confirm **Radius comp.: RL/RR/no comp?** by pressing the ENT key: Do not activate the radius compensation
- **Feed rate F=?** Enter the positioning feed rate, e.g. 3000 mm/min and confirm with the ENT key
- **Miscellaneous function M?** Switch on the spindle and coolant, e.g. **M13**. Confirm with the END key: The TNC saves the entered positioning block
- G 26 Move to the contour: Define the **rounding radius** of the approaching arc
- Machine the contour and move to contour point **2**: You only need to enter the information that changes. In other words, enter only the Y coordinate 95 and save your entry with the END key
- Move to contour point **3**: Enter the X coordinate 95 and save your entry with the END key
- Define the chamfer at contour point **3**: Enter the chamfer width 10 mm and save with the END key
- Move to contour point **4**: Enter the Y coordinate 5 and save your entry with the END key
- Define the chamfer at contour point **4**: Enter the chamfer width 20 mm and save with the END key
- Move to contour point **1**: Enter the X coordinate 5 and save your entry with the END key
- G 27 Depart the contour: Define the **rounding radius** of the departing arc
- G 0 Retract the tool: Press the orange axis key Z in order to get clear in the tool axis, and enter the value for the position to be approached, e.g. 250. Confirm with the ENT key
- Confirm **Radius comp.: RL/RR/no comp?** by pressing the ENT key: Do not activate the radius compensation
- **Miscellaneous function M?** Enter **M2** to end the program and confirm with the END key: The TNC saves the entered positioning block

Further information on this topic

- **Complete example with NC blocks**: See "Example: Linear movements and chamfers with Cartesian coordinates" on page 172
- Creating a new program: See "Creating and Writing Programs" on page 79
- Approaching/departing contours: See "Contour Approach and Departure" on page 159
- Programming contours: See "Overview of path functions" on page 163
- Tool radius compensation: See "Tool radius compensation" on page 151
- Miscellaneous functions (M): See "Miscellaneous Functions for Program Run Control, Spindle and Coolant" on page 271

Create a cycle program

The holes (depth of 20 mm) shown in the figure at right are to be drilled with a standard drilling cycle. You have already defined the workpiece blank.



TOOL CALL

- Call the tool: Enter the tool data. Confirm each of your entries with the ENT key. Do not forget the tool axis
- Press the L key to open a program block for a linear movement
- Press the left arrow key to switch to the input range for G codes

G00

- Press the G0 soft key if you want to enter a rapid traverse motion
- Retract the tool: Press the orange axis key Z in order to get clear in the tool axis, and enter the value for the position to be approached, e.g. 250. Confirm with the ENT key
- Confirm **Radius comp.: RL/RR/no comp?** by pressing the ENT key: Do not activate the radius compensation
- Confirm the **Miscellaneous function M?** with the END key: The TNC saves the entered positioning block

CYCL DEF

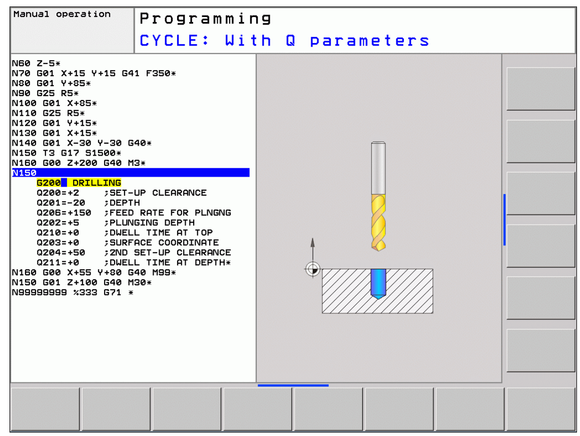
- Call the cycle menu

DRILLING/ THREAD

- Display the drilling cycles

200

- Select the standard drilling cycle 200: The TNC starts the dialog for cycle definition. Enter all parameters requested by the TNC step by step and conclude each entry with the ENT key. In the screen to the right, the TNC also displays a graphic showing the respective cycle parameter



G 0

- Move to the first drilling position: Enter the **coordinates** of the drilling position, switch on the coolant and spindle, and call the cycle with **M99**

X 0

- Move to the subsequent drilling positions: Enter the **coordinates** of the respective drilling positions, and call the cycle with **M99**

G 0

- Retract the tool: Press the orange axis key Z in order to get clear in the tool axis, and enter the value for the position to be approached, e.g. 250. Confirm with the ENT key
- Confirm **Radius comp.: RL/RR/no comp?** by pressing the ENT key: Do not activate the radius compensation
- **Miscellaneous function M?** Enter **M2** to end the program and confirm with the END key: The TNC saves the entered positioning block

Example NC blocks

%C200 G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	Definition of workpiece blank
N20 G31 X+100 Y+100 Z+0 *	
N30 T5 G17 S4500 *	Tool call
N40 G00 G40 G90 Z+250 *	Retract the tool
N50 G200 DRILLING	Define the cycle
Q200=2 ;SETUP CLEARANCE	
Q201=-20 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGN	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=-10 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q211=0.2 ;DWELL TIME AT DEPTH	
N60 X+10 Y+10 M13 M99 *	Spindle and coolant on, call cycle
N70 X+10 Y+90 M99 *	Call the cycle
N80 X+90 Y+10 M99 *	Call the cycle
N90 X+90 Y+90 M99 *	Call the cycle
N100 G00 Z+250 M2 *	Retract in the tool axis, end program
N99999999 %C200 G71 *	

Further information on this topic

- Creating a new program: See "Creating and Writing Programs" on page 79
- Cycle programming: See User’s Manual for Cycles

1.4 Graphically Testing the First Part

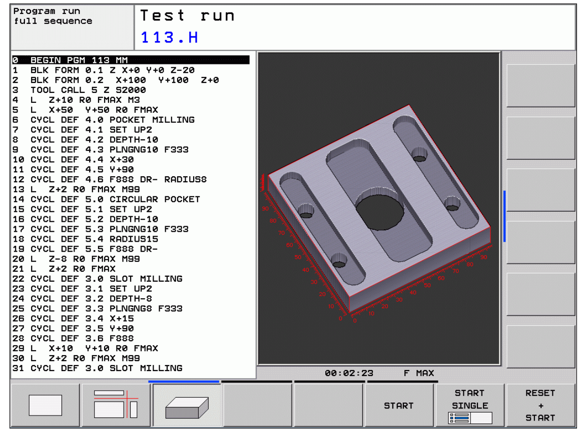
Select the correct operating mode

You can test programs only in the Test Run mode:

▶ Press the operating modes key: The TNC goes into the **Test Run** mode

Further information on this topic

- Operating modes of the TNC: See "Operating Modes" on page 58
- Testing programs: See "Test Run" on page 379



Select the tool table for the test run

You only need to execute this step if you have not activated a tool table in the Test Run mode.

▶ Press the PGM MGT key: the TNC displays the file management

▶ Press the SELECT TYPE soft key: The TNC shows a soft-key menu for selection of the file type to be displayed

▶ Press the SHOW ALL soft key: The TNC shows all saved files in the right window

▶ Move the highlight to the left onto the directories

▶ Move the highlight to the **TNC:** directory

▶ Move the highlight to the right onto the files

▶ Move the highlight to the file TOOL.T (active tool table) and load with the ENT key: TOOL.T receives the status **S** and is therefore active for the Test Run

▶ Press the END key: Leave the file manager

Further information on this topic

- Tool management: See "Entering tool data in the table" on page 134
- Testing programs: See "Test Run" on page 379

Choose the program you want to test

- Press the PGM MGT key: The TNC displays the file manager.
- Press the LAST FILES soft key: The TNC opens a pop-up window with the most recently selected files
- Use the arrow keys to select the program that you want to test. Load with the ENT key

Further information on this topic

- Selecting a program: See "Working with the File Manager" on page 92

Select the screen layout and the view

- Press the key for selecting the screen layout. The TNC shows all available alternatives in the soft-key row
- Press the PROGRAM + GRAPHICS soft key: In the left half of the screen the TNC shows the program; in the right half it shows the workpiece blank
- Select the desired view via soft key
- Plan view
- Projection in three planes
- 3-D view

Further information on this topic

- Graphic functions: See "Graphics" on page 368
- Running a test run: See "Test Run" on page 379

Start the program test

- Press the RESET + START soft key: The TNC simulates the active program up to a programmed break or to the program end
- While the simulation is running you can use the soft keys to change views.
- Press the STOP soft key: The TNC interrupts the test run
- Press the START soft key: The TNC resumes the test run after a break

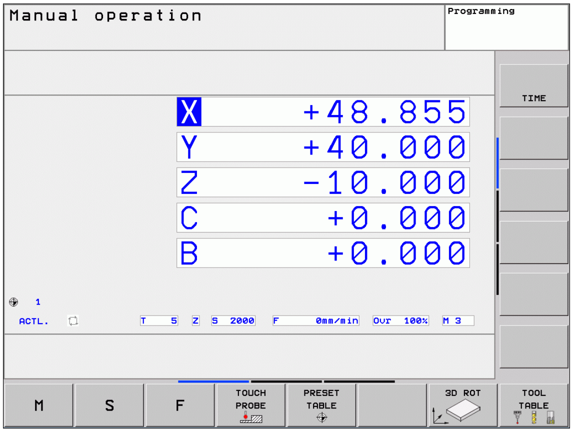
Further information on this topic

- Running a test run: See "Test Run" on page 379
- Graphic functions: See "Graphics" on page 368

1.5 Tool Setup

Select the correct operating mode

Tools are set up in the **Manual Operation** mode:



- ▶ Press the operating modes key: The TNC goes into the **Manual Operation** mode

Further information on this topic

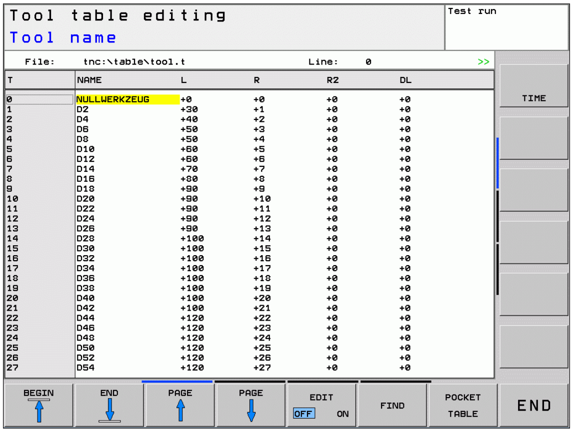
- Operating modes of the TNC: See "Operating Modes" on page 58

Prepare and measure tools

- ▶ Clamp the required tools in their chucks
- ▶ When measuring with an external tool presetter: Measure the tools, note down the length and radius, or transfer them directly to the machine through a transfer program
- ▶ When measuring on the machine: Place the tools into the tool changer (see page 48)

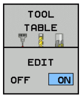
The tool table TOOL.T

In the tool table TOOL.T (permanently saved under **TNC:\TABLE**), save the tool data such as length and radius, but also further tool-specific information that the TNC needs to conduct its functions.



T	NAME	L	R	R2	DL
0	NULLWERKZEUG	+0	+0	+0	+0
1	D2	+30	+1	+0	+0
2	D4	+40	+2	+0	+0
3	D6	+50	+3	+0	+0
4	D8	+50	+4	+0	+0
5	D10	+60	+5	+0	+0
6	D12	+60	+6	+0	+0
7	D14	+70	+7	+0	+0
8	D16	+80	+8	+0	+0
9	D18	+90	+9	+0	+0
10	D20	+90	+10	+0	+0
11	D22	+90	+11	+0	+0
12	D24	+90	+12	+0	+0
13	D26	+90	+13	+0	+0
14	D28	+100	+14	+0	+0
15	D30	+100	+15	+0	+0
16	D32	+100	+16	+0	+0
17	D34	+100	+17	+0	+0
18	D36	+100	+18	+0	+0
19	D38	+100	+19	+0	+0
20	D40	+100	+20	+0	+0
21	D42	+100	+21	+0	+0
22	D44	+120	+22	+0	+0
23	D46	+120	+23	+0	+0
24	D48	+120	+24	+0	+0
25	D50	+120	+25	+0	+0
26	D52	+120	+26	+0	+0
27	D54	+120	+27	+0	+0

To enter tool data in the tool table TOOL.T, proceed as follows:



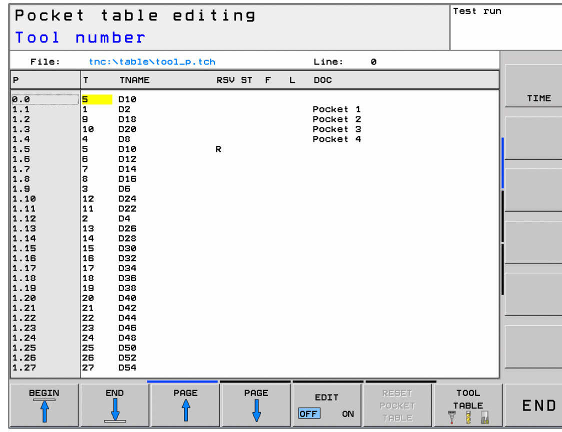
- ▶ Display the tool table
- ▶ Edit the tool table: Set the EDITING soft key to ON
- ▶ With the upward or downward arrow keys you can select the tool number that you want to edit
- ▶ With the rightward or leftward arrow keys you can select the tool data that you want to edit
- ▶ To leave the tool table, press the END key

Further information on this topic

- Operating modes of the TNC: See "Operating Modes" on page 58
- Working with the tool table: See "Entering tool data in the table" on page 134

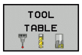
The pocket table TOOL_P.CH

The function of the pocket table depends on the machine. Your machine manual provides more detailed information.

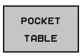


In the pocket table TOOL_P.TCH (permanently saved under **TNC:\TABLE**) you specify which tools your tool magazine contains.

To enter data in the pocket table TOOL_P.TCH, proceed as follows:



▶ Display the tool table



▶ Display the pocket table

▶ Edit the pocket table: Set the EDITING soft key to ON

▶ With the upward or downward arrow keys you can select the pocket number that you want to edit

▶ With the rightward or leftward arrow keys you can select the data that you want to edit

▶ To leave the pocket table, press the END key

Further information on this topic

- Operating modes of the TNC: See "Operating Modes" on page 58
- Working with the pocket table: See "Pocket table for tool changer" on page 141

1.6 Workpiece Setup

Select the correct operating mode

Workpieces are set up in the **Manual Operation** or **Electronic Handwheel** mode

▶ Press the operating modes key: The TNC goes into the **Manual Operation** mode

Further information on this topic

- Manual mode: See "Moving the Machine Axes" on page 329

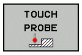
Clamp the workpiece

Mount the workpiece with a fixture on the machine table. If you have a 3-D touch probe on your machine, then you do not need to clamp the workpiece parallel to the axes.

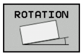
If you do not have a 3-D touch probe available, you have to align the workpiece so that it is fixed with its edges parallel to the machine axes.

Align the workpiece with a 3-D touch probe system

- Insert the 3-D touch probe: In the Manual Data Input (MDI) operating mode, run a **TOOL CALL** block containing the tool axis, and then return to the **Manual Operation** mode (in MDI mode you can run an individual NC block independently of the others)



- Select the probing functions: The TNC displays the available functions in the soft-key row



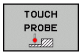
- Measure the basic rotation: The TNC displays the basic rotation menu. To identify the basic rotation, probe two points on a straight surface of the workpiece
- Use the axis-direction keys to pre-position the touch probe to a position near the first contact point
- Select the probing direction via soft key
- Press NC start: The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point
- Use the axis-direction keys to pre-position the touch probe to a position near the second contact point
- Press NC start: The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point
- Then the TNC shows the measured basic rotation
- Press SET BASIC ROTATION soft key to select the displayed value as the active rotation. Press the END soft key to exit the menu

Further information on this topic

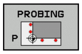
- MDI operating mode: See "Programming and Executing Simple Machining Operations" on page 362
- Workpiece alignment: See "Compensating Workpiece Misalignment with a 3-D Touch Probe" on page 348

Set the datum with a 3-D touch probe

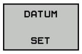
- Insert the 3-D touch probe: In the MDI mode, run a **TOOL CALL** block containing the tool axis and then return to the **Manual Operation** mode



- Select the probing functions: The TNC displays the available functions in the soft-key row



- Set the datum at a workpiece corner, for example
- Position the touch probe near the first touch point on the first workpiece edge
- Select the probing direction via soft key
- Press NC start: The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point
- Use the axis-direction keys to pre-position the touch probe to a position near the second touch point on the first workpiece edge
- Press NC start: The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point
- Use the axis-direction keys to pre-position the touch probe to a position near the first touch point on the second workpiece edge
- Select the probing direction via soft key
- Press NC start: The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point
- Use the axis-direction keys to pre-position the touch probe to a position near the second touch point on the second workpiece edge
- Press NC start: The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point
- Then the TNC shows the coordinates of the measured corner point



- Set to 0: Press the SET DATUM soft key
- Press the END soft key to close the menu

Further information on this topic

- Datum setting: See "Set the datum with a 3-D touch probe" on page 350

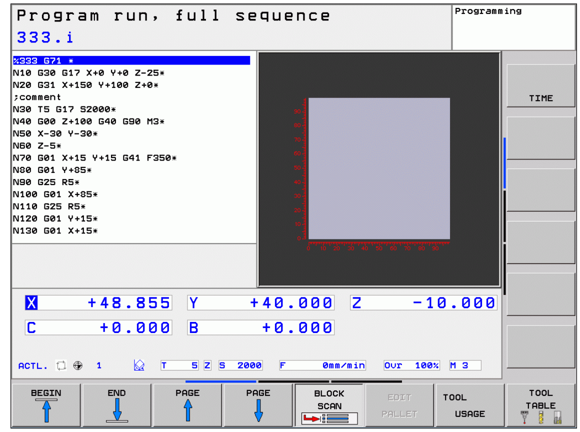
1.7 Running the First Program

Select the correct operating mode

You can run programs either in the Single Block or the Full Sequence mode:

▶ Press the operating mode key: The TNC goes into the **Program Run, Single Block** mode and the TNC executes the program block by block. You have to confirm each block with the NC key

▶ Press the operating mode key: The TNC goes into the **Program Run, Full Sequence** mode and the TNC executes the program after NC start up to a program break or to the end of the program



Further information on this topic

- Operating modes of the TNC: See "Operating Modes" on page 58
- Running programs: See "Program Run" on page 382

Choose the program you want to run

PGM MGT

▶ Press the PGM MGT key: the TNC displays the file management

LAST FILES

▶ Press the LAST FILES soft key: The TNC opens a pop-up window with the most recently selected files

▶ If desired, use the arrow keys to select the program that you want to run. Load with the ENT key

Further information on this topic

- File management: See "Working with the File Manager" on page 92

Start the program

I

▶ Press the NC start button: The TNC executes the active program

Further information on this topic

- Running programs: See "Program Run" on page 382

2

Introduction

2.1 The TNC 320

HEIDENHAIN TNC controls are workshop-oriented contouring controls that enable you to program conventional machining operations right at the machine in an easy-to-use conversational programming language. They are designed for milling and drilling machine tools, as well as machining centers, with up to five axes. You can also change the angular position of the spindle under program control.

Keyboard and screen layout are clearly arranged in such a way that the functions are fast and easy to use.

Programming: HEIDENHAIN conversational and ISO formats

The HEIDENHAIN conversational programming format is an especially easy method of writing programs. Interactive graphics illustrate the individual machining steps for programming the contour. If a production drawing is not dimensioned for NC, the HEIDENHAIN FK free contour programming performs the necessary calculations automatically. Workpiece machining can be graphically simulated either during or before actual machining.

It is also possible to program the TNCs in ISO format or DNC mode.

You can also enter and test one program while the control is running another.

Compatibility

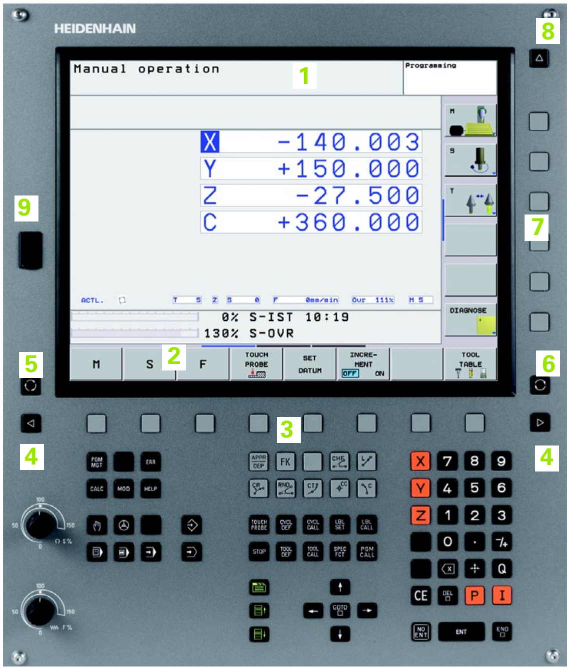
The scope of functions of the TNC 320 does not correspond to that of the TNC 4xx and iTNC 530 series of controls. Therefore, machining programs created on HEIDENHAIN contouring controls (starting from the TNC 150 B) may not always run on the TNC 320. If NC blocks contain invalid elements, the TNC will mark them as ERROR blocks when the file is opened.

Please also note the detailed description of the differences between the iTNC 530 and the TNC 320 (see "Comparison: Functions of the TNC 320 and the iTNC 530" on page 435).

2.2 Visual Display Unit and Keyboard

Visual display unit

The TNC is shipped with a 15-inch TFT flat-panel display.



1 Header

When the TNC is on, the selected operating modes are shown in the screen header: the machining mode at the left and the programming mode at right. The currently active mode is displayed in the larger box, where the dialog prompts and TNC messages also appear (unless the TNC is showing only graphics).

2 Soft keys

In the footer the TNC indicates additional functions in a soft-key row. You can select these functions by pressing the keys immediately below them. The lines immediately above the soft-key row indicate the number of soft-key rows that can be called with the black arrow keys to the right and left. The active soft-key row is indicated by brightened bar.

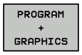
3 Soft-key selection keys
4 Shift between soft-key rows
5 Setting the screen layout
6 Shift key for switchover between machining and programming modes
7 Soft-key selection keys for machine tool builders
8 Switches soft-key rows for machine tool builders
9 USB connection

Setting the screen layout

You select the screen layout yourself: In the PROGRAMMING AND EDITING mode of operation, for example, you can have the TNC show program blocks in the left window while the right window displays programming graphics. You could also display the program structure in the right window instead, or display only program blocks in one large window. The available screen windows depend on the selected operating mode.

To change the screen layout:

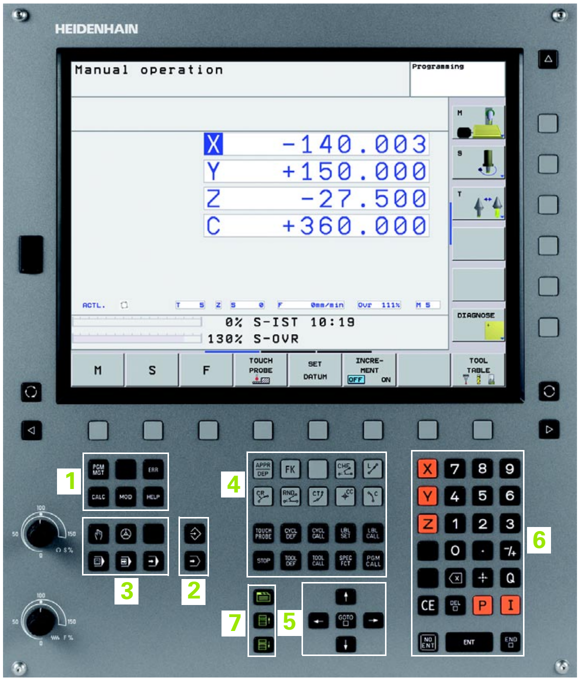
Press the SPLIT SCREEN key: The soft-key row shows the available layout options (see "Operating Modes", page 58).



Select the desired screen layout.

Operating panel

The TNC 320 is delivered with an integrated keyboard. The figure at right shows the controls and displays of the keyboard:



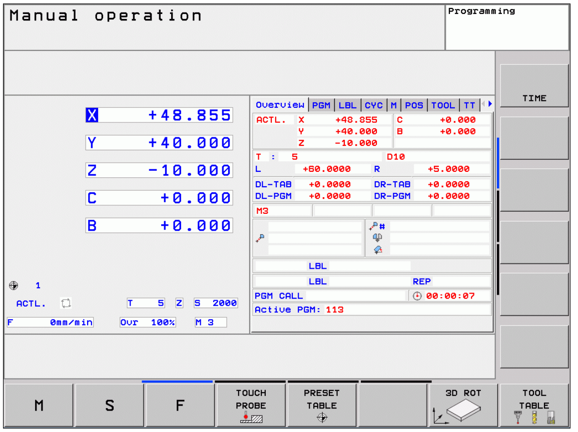
- **1**
  - File management
  - Calculator
  - MOD function
  - HELP function
- **2** Programming modes
- **3** Machine operating modes
- **4** Initiation of programming dialog
- **5** Arrow keys and GOTO jump command
- **6** Numerical input and axis selection
- **7** Navigation keys

The functions of the individual keys are described on the inside front cover.

> Machine panel buttons, e.g. NC START or NC STOP, are described in the manual for your machine tool.

2.3 Operating Modes

Manual Operation and Electronic Handwheel



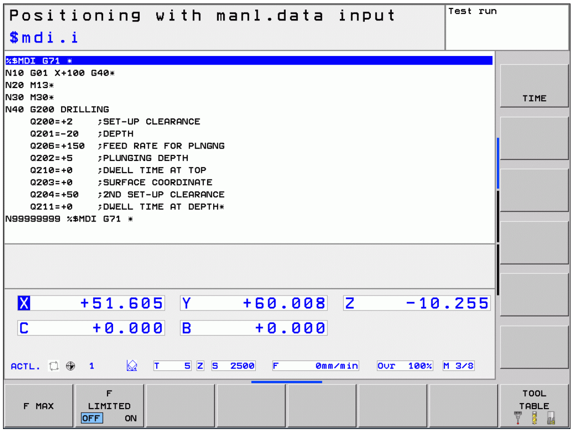
The Manual Operation mode is required for setting up the machine tool. In this mode of operation, you can position the machine axes manually or by increments, set the datums, and tilt the working plane.

The Electronic Handwheel mode of operation allows you to move the machine axes manually with the HR electronic handwheel.

Soft keys for selecting the screen layout (select as described previously)

Window	Soft key
Positions	POSITION
Left: positions, right: status display	POSITION + STATUS

Positioning with Manual Data Input



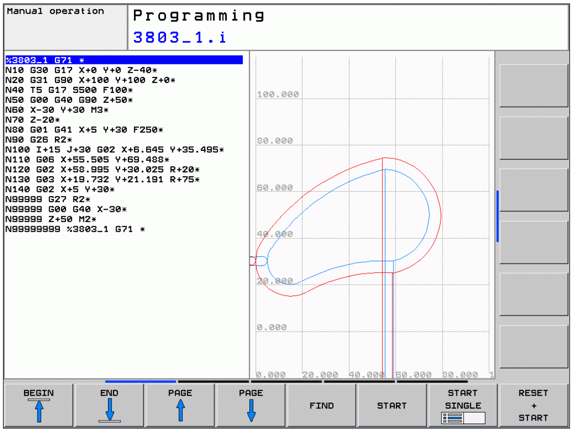
This mode of operation is used for programming simple traversing movements, such as for face milling or pre-positioning.

Soft keys for selecting the screen layout

Window	Soft key
Program	PGM
Left: program blocks, right: status display	PROGRAM + STATUS

Programming and Editing

In this mode of operation you can write your part programs. The FK free programming feature, the various cycles and the Q parameter functions help you with programming and add necessary information. If desired, you can have the programming graphics show the programmed paths of traverse.

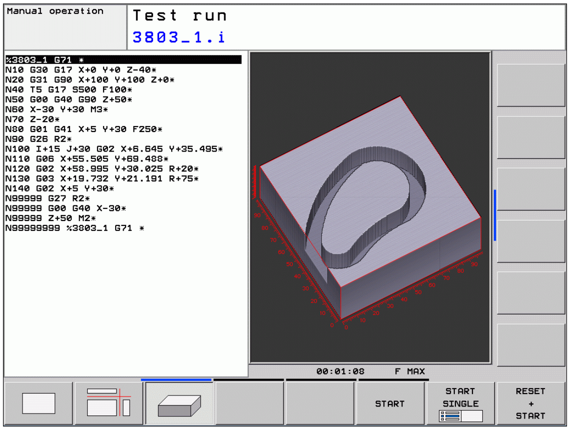


Soft keys for selecting the screen layout

Window	Soft key
Program	PGM
Left: program, right: program structure	PROGRAM + SECTS
Left: program blocks, right: graphics	PROGRAM + GRAPHICS

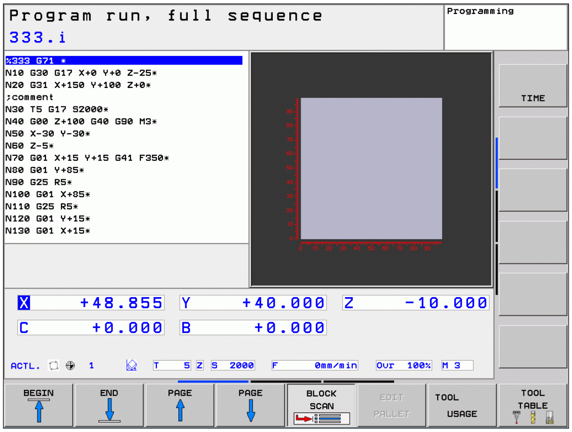
Test Run

In the Test Run mode of operation, the TNC checks programs and program sections for errors, such as geometrical incompatibilities, missing or incorrect data within the program or violations of the work space. This simulation is supported graphically in different display modes.



Soft keys for selecting the screen layout: see "Program Run, Full Sequence and Program Run, Single Block", page 60.

Program Run, Full Sequence and Program Run, Single Block



In the Program Run, Full Sequence mode of operation the TNC executes a part program continuously to its end or to a manual or programmed stop. You can resume program run after an interruption.

In the Program Run, Single Block mode of operation you execute each block separately by pressing the machine START button.

Soft keys for selecting the screen layout

Window	Soft key
Program	PGM
Left: program, right: program structure	PROGRAM + SECTS
Left: program, right: status	PROGRAM + STATUS
Left: program blocks, right: graphics	PROGRAM + GRAPHICS
Graphics	GRAPHICS

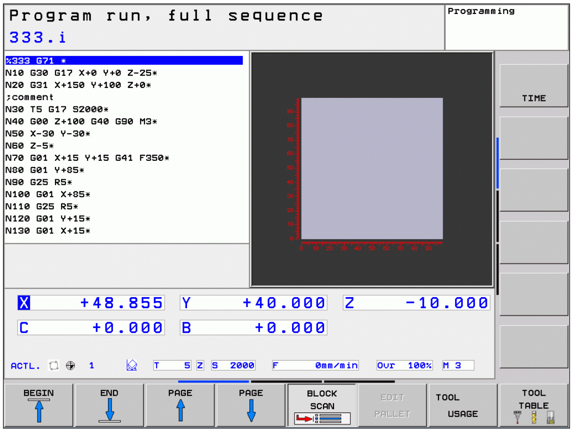
2.4 Status Displays

"General" status display

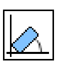
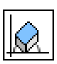
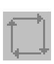
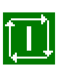
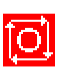
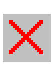
The general status display in the lower part of the screen informs you of the current state of the machine tool. It is displayed automatically in the following modes of operation:

- Program Run, Single Block and Program Run, Full Sequence, except if the screen layout is set to display graphics only, and
- Positioning with Manual Data Input (MDI).

In the Manual mode and Electronic Handwheel mode the status display appears in the large window.



Information in the status display

Symbol	Meaning
ACTL.	Actual or nominal coordinates of the current position
XYZ	Machine axes; the TNC displays auxiliary axes in lower-case letters. The sequence and quantity of displayed axes is determined by the machine tool builder. Refer to your machine manual for more information.
FSM	The displayed feed rate in inches corresponds to one tenth of the effective value. Spindle speed S, feed rate F and active M functions.
*	Program run started
	Axis is locked
	Axis can be moved with the handwheel
	Axes are moving under a basic rotation
	Axes are moving in a tilted working plane
	No active program
	Program run has started
	Program run is stopped
	Program run is being aborted

Additional status displays

The additional status displays contain detailed information on the program run. They can be called in all operating modes except for the Programming and Editing mode of operation.

To switch on the additional status display:

Call the soft-key row for screen layout.

PROGRAM + STATUS — Screen layout with additional status display: In the right half of the screen, the TNC shows the **Overview** status form.

To select an additional status display:

Shift the soft-key rows until the STATUS soft keys appear.

STATUS POS. — Either select the additional status display, e.g. positions and coordinates, or

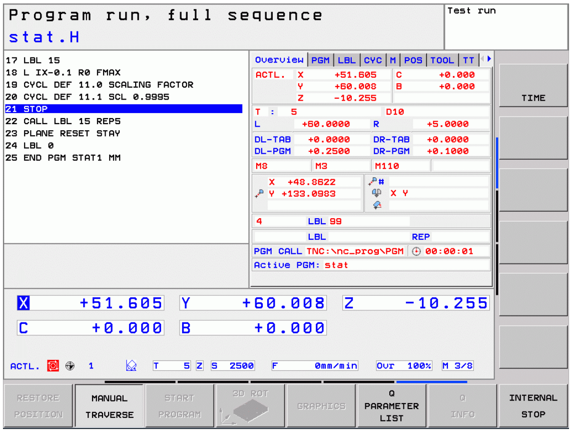
use the soft keys to select the desired view.

With the soft keys or switch-over soft keys, you can choose directly between the available status displays.

Please note that some of the status information described below is not available unless the associated software option is enabled on your TNC.

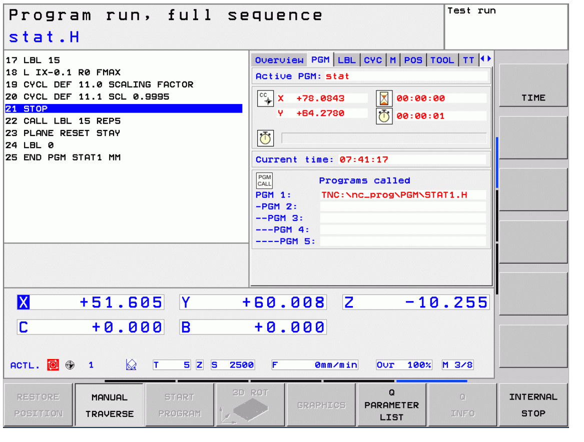
Overview

After switch-on, the TNC displays the **Overview** status form, provided that you have selected the PROGRAM+STATUS screen layout (or POSITION + STATUS). The overview form contains a summary of the most important status information, which you can also find on the various detail forms.



Soft key	Meaning
STATUS OVERVIEW	Position display
	Tool information
	Active M functions
	Active coordinate transformations
	Active subprogram
	Active program section repeat
	Program called with **PGM CALL**
	Current machining time
	Name of the active main program

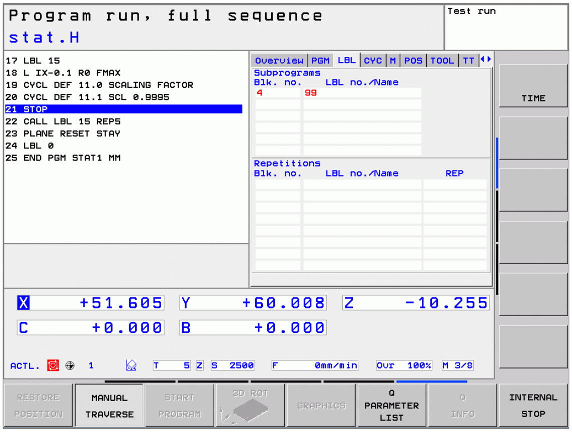
General program information (PGM tab)



Soft key	Meaning
No direct selection possible	Name of the active main program
	Circle center CC (pole)
	Dwell time counter
	Machining time when the program was completely simulated in the **Test Run** operating mode
	Current machining time in percent
	Current time
	Active programs

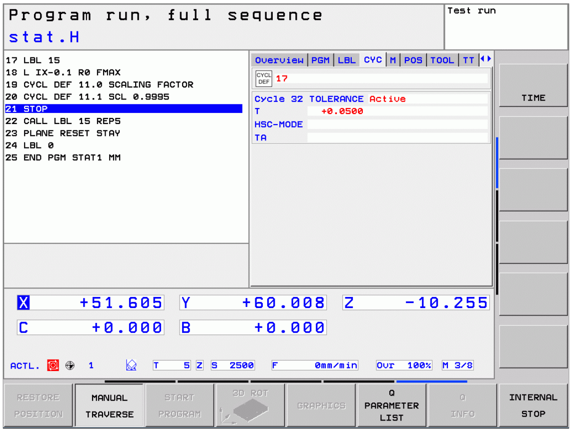
Program section repeat/Subprograms (LBL tab)

Soft key	Meaning
No direct selection possible	Active program section repeats with block number, label number, and number of programmed repeats/repeats yet to be run
	Active subprogram numbers with block number in which the subprogram was called and the label number that was called



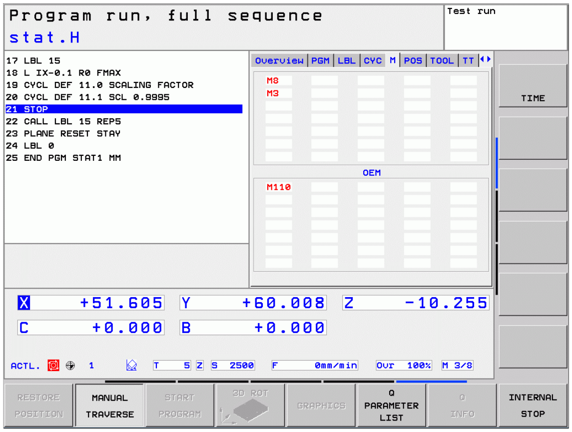
Information on standard cycles (CYC tab)

Soft key	Meaning
No direct selection possible	Active machining cycle
	Active values of Cycle G62 Tolerance



Active miscellaneous functions M (M tab)

Soft key	Meaning
No direct selection possible	List of the active M functions with fixed meaning
	List of the active M functions that are adapted by your machine manufacturer

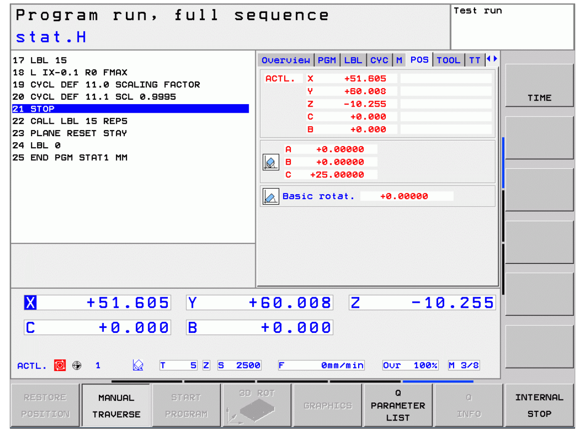


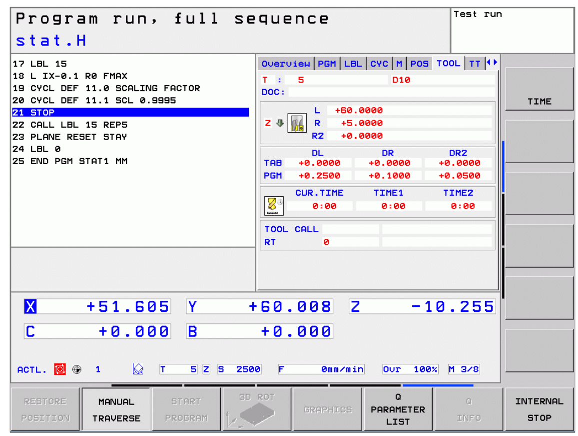
Positions and coordinates (POS tab)

Soft key	Meaning
STATUS POS.	Type of position display, e.g. actual position
	Tilt angle of the working plane
	Angle of a basic rotation

Information on tools (TOOL tab)

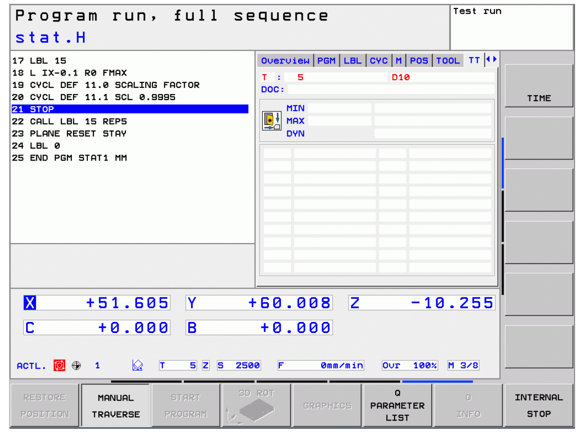
Soft key	Meaning
TOOL STATUS	■ T: Tool number and name ■ RT: Number and name of a replacement tool
	Tool axis
	Tool lengths and radii
	Oversizes (delta values) from the tool table (TAB) and the **TOOL CALL** (PGM)
	Tool life, maximum tool life (TIME 1) and maximum tool life for **TOOL CALL** (TIME 2)
	Display of the active tool and the (next) replacement tool





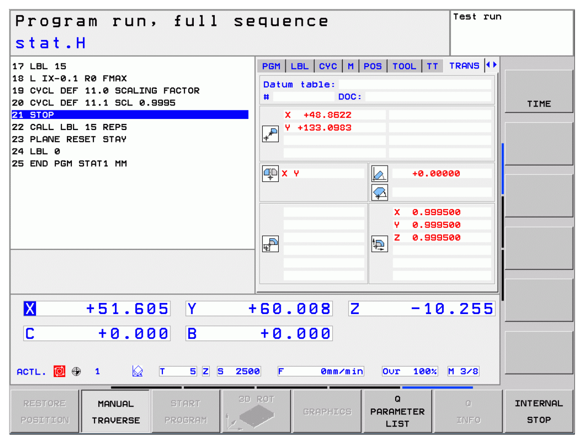
Tool measurement (TT tab)

> The TNC displays the TT tab only if the function is active on your machine.



Soft key	Meaning
No direct selection possible	Number of the tool to be measured
	Display whether the tool radius or the tool length is being measured
	MIN and MAX values of the individual cutting edges and the result of measuring the rotating tool (DYN = dynamic measurement)
	Cutting edge number with the corresponding measured value. If the measured value is followed by an asterisk, the allowable tolerance in the tool table was exceeded

Coordinate transformations (TRANS tab)

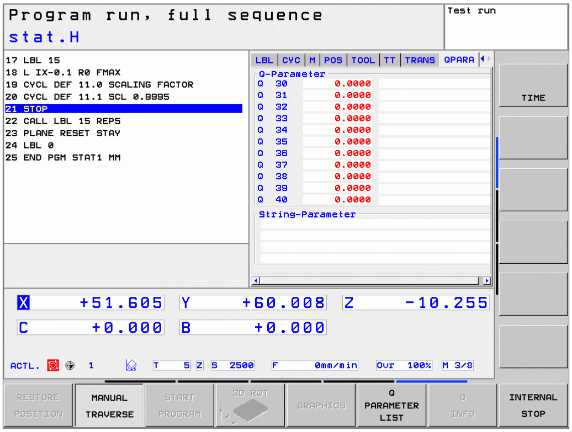


Soft key	Meaning
STATUS COORD. TRANSF.	Name of the active datum table
	Active datum number (#), comment from the active line of the active datum number (**DOC**) from Cycle G53
	Active datum shift (Cycle G54); The TNC displays an active datum shift in up to 8 axes
	Mirrored axes (Cycle G28)
	Active basic rotation
	Active rotation angle (Cycle G73)
	Active scaling factor/factors (Cycles G72); The TNC displays an active scaling factor in up to 6 axes
	Scaling datum

For further information, refer to the User's Manual for Cycles, "Coordinate Transformation Cycles."

Display Q parameters (QPARA tab)

Soft key	Meaning
STATUS OF Q PARAM.	Display the current values of the defined Q parameters
	Display the character strings of the defined string parameters



Press the Q PARAMETER LIST soft key. The TNC opens a pop-up window in which you can enter the desired range for display of the Q parameters or string parameters. Multiple Q parameters are entered separated by commas (e.g. Q 1,2,3,4). To define display ranges, enter a hyphen (e.g. Q 10-14).

2.5 Accessories: HEIDENHAIN 3-D Touch Probes and Electronic Handwheels

3-D touch probes

With the various HEIDENHAIN 3-D touch probe systems you can:

- Automatically align workpieces
- Quickly and precisely set datums
- Measure the workpiece during program run
- Measure and inspect tools

All of the touch probe functions are described in the User's Manual for Cycle Programming. Please contact HEIDENHAIN if you need a copy of this User's Manual. ID: 679 220-xx.

TS 220, TS 440, TS 444, TS 640 und TS 740 touch trigger probes

These touch probes are particularly effective for automatic workpiece alignment, datum setting and workpiece measurement. The TS 220 transmits the triggering signals to the TNC via cable and is a cost-effective alternative for applications where digitizing is not frequently required.

The TS 640 (see figure) and the smaller TS 440 feature infrared transmission of the triggering signal to the TNC. This makes them highly convenient for use on machines with automatic tool changers.

Principle of operation: HEIDENHAIN triggering touch probes feature a wear resisting optical switch that generates an electrical signal as soon as the stylus is deflected. This signal is transmitted to the control, which stores the current position of the stylus as the actual value.

TT 140 tool touch probe for tool measurement

The TT 140 is a triggering 3-D touch probe for tool measurement and inspection. Your TNC provides three cycles for this touch probe with which you can measure the tool length and radius automatically either with the spindle rotating or stopped. The TT 140 features a particularly rugged design and a high degree of protection, which make it insensitive to coolants and swarf. The triggering signal is generated by a wear-resistant and highly reliable optical switch.

HR electronic handwheels

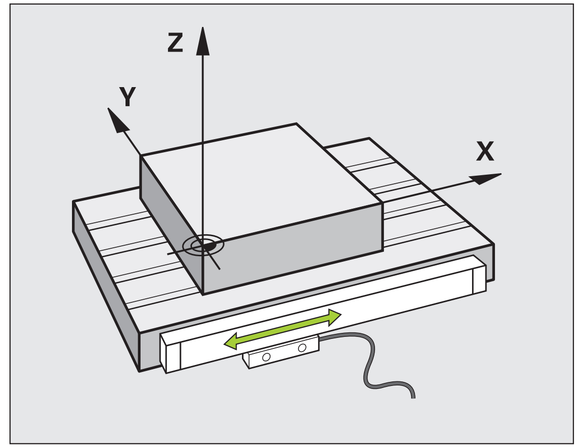
Electronic handwheels facilitate moving the axis slides precisely by hand. A wide range of traverses per handwheel revolution is available. Apart from the HR 130 and HR 150 integral handwheels, HEIDENHAIN also offers the HR 410 portable handwheel.

3

Programming: Fundamentals, File Management

3.1 Fundamentals

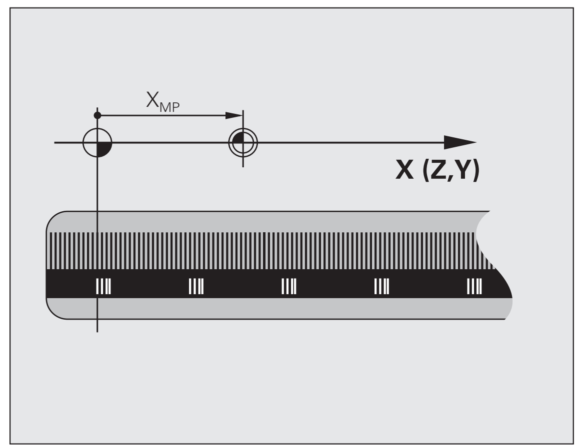
Position encoders and reference marks



The machine axes are equipped with position encoders that register the positions of the machine table or tool. Linear axes are usually equipped with linear encoders, rotary tables and tilting axes with angle encoders.

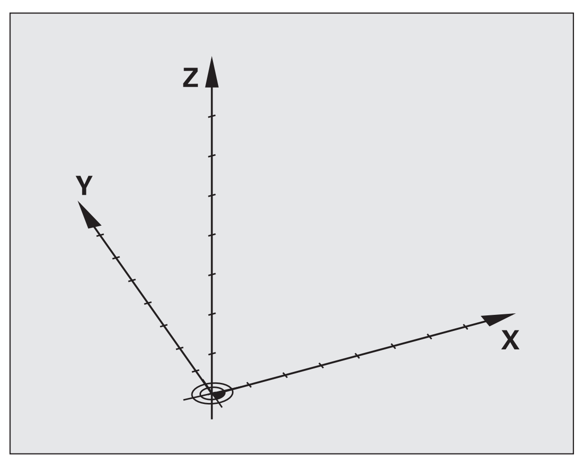
When a machine axis moves, the corresponding position encoder generates an electrical signal. The TNC evaluates this signal and calculates the precise actual position of the machine axis.

If there is a power interruption, the calculated position will no longer correspond to the actual position of the machine slide. To recover this association, incremental position encoders are provided with reference marks. The scales of the position encoders contain one or more reference marks that transmit a signal to the TNC when they are crossed over. From that signal the TNC can re-establish the assignment of displayed positions to machine positions. For linear encoders with distance-coded reference marks, the machine axes need to move by no more than 20 mm, for angle encoders by no more than 20°.



With absolute encoders, an absolute position value is transmitted to the control immediately upon switch-on. In this way the assignment of the actual position to the machine slide position is re-established directly after switch-on.

Reference system



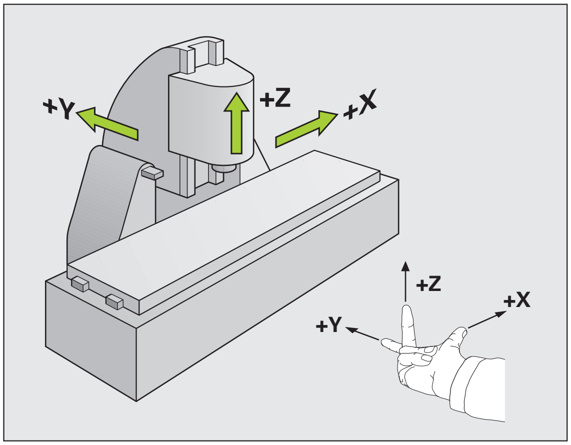
A reference system is required to define positions in a plane or in space. The position data are always referenced to a predetermined point and are described through coordinates.

The Cartesian coordinate system (a rectangular coordinate system) is based on the three coordinate axes X, Y and Z. The axes are mutually perpendicular and intersect at one point called the datum. A coordinate identifies the distance from the datum in one of these directions. A position in a plane is thus described through two coordinates, and a position in space through three coordinates.

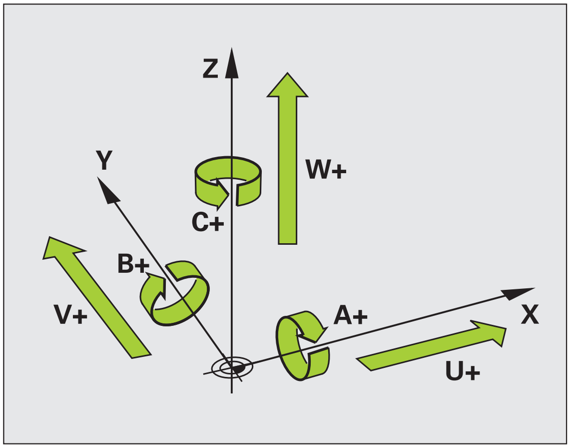
Coordinates that are referenced to the datum are referred to as absolute coordinates. Relative coordinates are referenced to any other known position (reference point) you define within the coordinate system. Relative coordinate values are also referred to as incremental coordinate values.

Reference system on milling machines

When using a milling machine, you orient tool movements to the Cartesian coordinate system. The illustration at right shows how the Cartesian coordinate system describes the machine axes. The figure illustrates the right-hand rule for remembering the three axis directions: the middle finger points in the positive direction of the tool axis from the workpiece toward the tool (the Z axis), the thumb points in the positive X direction, and the index finger in the positive Y direction.



The TNC 320 can control up to 5 axes optionally. The axes U, V and W are secondary linear axes parallel to the main axes X, Y and Z, respectively. Rotary axes are designated as A, B and C. The illustration at lower right shows the assignment of secondary axes and rotary axes to the main axes.

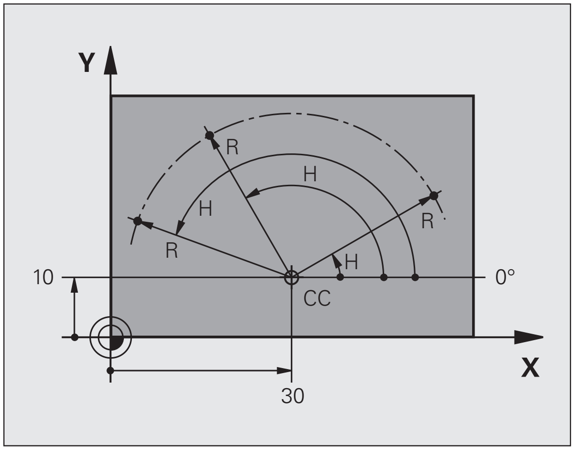


Designation of the axes on milling machines

The X, Y and Z axes on your milling machine are also referred to as tool axis, principal axis (1st axis) and minor axis (2nd axis). The assignment of the tool axis is decisive for the assignment of the principal and minor axes.

Tool axis	Principal axis	Minor axis
X	Y	Z
Y	Z	X
Z	X	Y

Polar coordinates



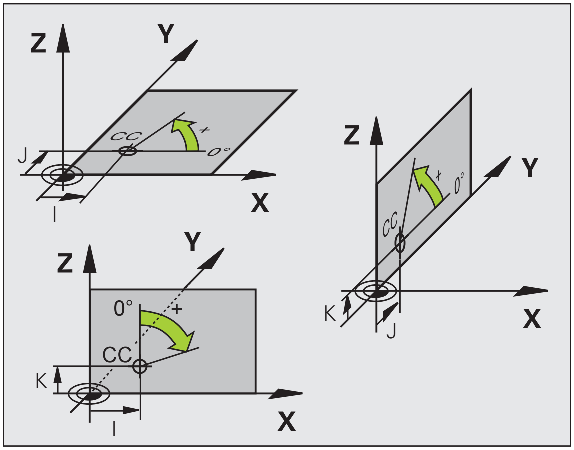
If the production drawing is dimensioned in Cartesian coordinates, you also write the NC program using Cartesian coordinates. For parts containing circular arcs or angles it is often simpler to give the dimensions in polar coordinates.

While the Cartesian coordinates X, Y and Z are three-dimensional and can describe points in space, polar coordinates are two-dimensional and describe points in a plane. Polar coordinates have their datum at a circle center (CC), or pole. A position in a plane can be clearly defined by the:

- Polar Radius, the distance from the circle center CC to the position, and the
- Polar Angle, the value of the angle between the reference axis and the line that connects the circle center CC with the position.

Setting the pole and the angle reference axis

The pole is set by entering two Cartesian coordinates in one of the three planes. These coordinates also set the reference axis for the polar angle H.

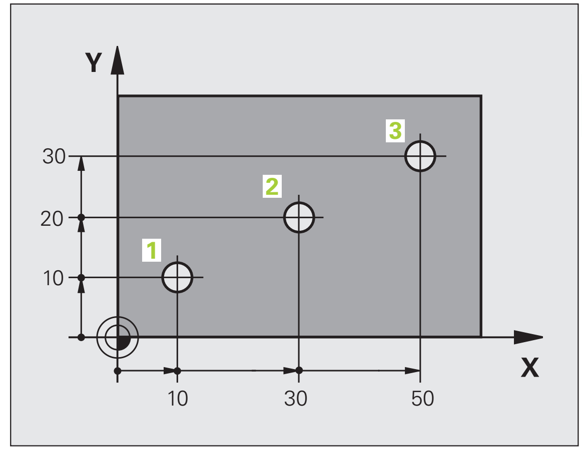


Coordinates of the pole (plane)	Reference axis of the angle
X/Y	+X
Y/Z	+Y
Z/X	+Z

Absolute and incremental workpiece positions

Absolute workpiece positions

Absolute coordinates are position coordinates that are referenced to the datum of the coordinate system (origin). Each position on the workpiece is uniquely defined by its absolute coordinates.

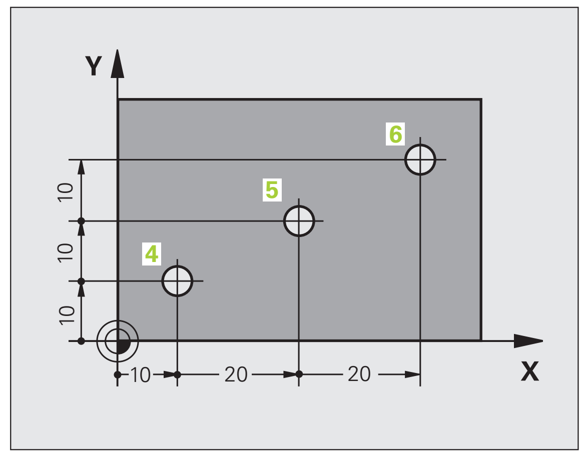


Example 1: Holes dimensioned in absolute coordinates

Hole **1**	Hole **2**	Hole **3**
X = 10 mm	X = 30 mm	X = 50 mm
Y = 10 mm	Y = 20 mm	Y = 30 mm

Incremental workpiece positions

Incremental coordinates are referenced to the last programmed nominal position of the tool, which serves as the relative (imaginary) datum. When you write an NC program in incremental coordinates, you thus program the tool to move by the distance between the previous and the subsequent nominal positions. This is why they are also referred to as a chain dimensions.



To program a position in incremental coordinates, enter the function G91 before the axis.

Example 2: Holes dimensioned in incremental coordinates

Absolute coordinates of hole **4**

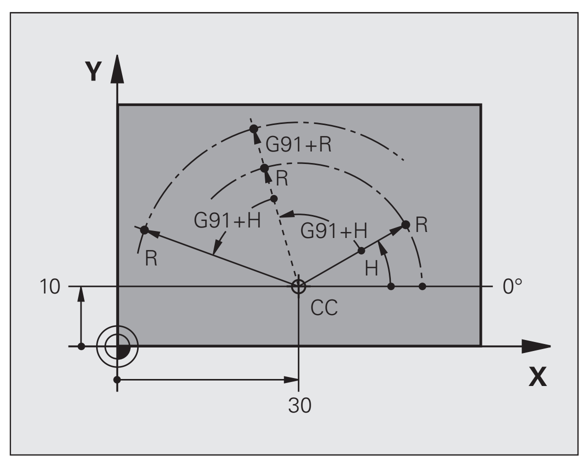
X = 10 mm
Y = 10 mm

Hole **5**, with respect to **4**	Hole **6**, with respect to **5**
G91 X = 20 mm	G91 X = 20 mm
G91 Y = 10 mm	G91 Y = 10 mm

Absolute and incremental polar coordinates

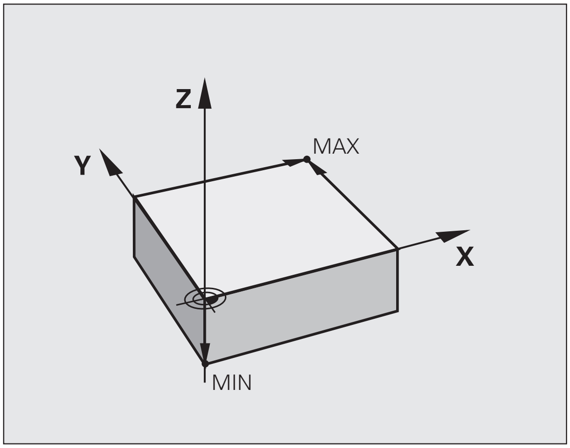
Absolute polar coordinates always refer to the pole and the reference axis.

Incremental polar coordinates always refer to the last programmed nominal position of the tool.



Setting the datum

A production drawing identifies a certain form element of the workpiece, usually a corner, as the absolute datum. When setting the datum, you first align the workpiece along the machine axes, and then move the tool in each axis to a defined position relative to the workpiece. Set the display of the TNC either to zero or to a known position value for each position. This establishes the reference system for the workpiece, which will be used for the TNC display and your part program.



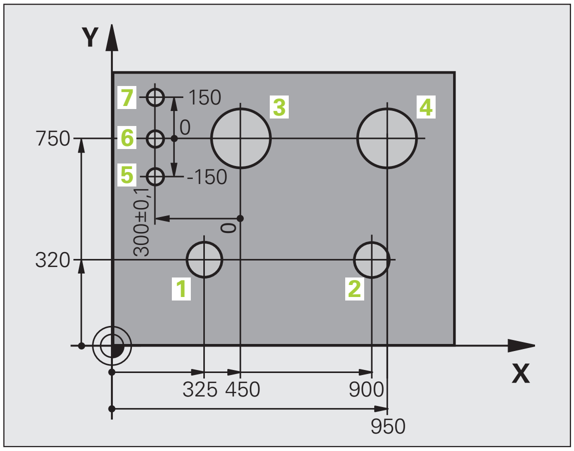
If the production drawing is dimensioned in relative coordinates, simply use the coordinate transformation cycles (see User's Manual for Cycles, Cycles for Coordinate Transformation).

If the production drawing is not dimensioned for NC, set the datum at a position or corner on the workpiece from which the dimensions of the remaining workpiece positions can be most easily measured.

The fastest, easiest and most accurate way of setting the datum is by using a 3-D touch probe from HEIDENHAIN. See "Setting the Datum with a 3-D Touch Probe" in the Touch Probe Cycles User's Manual.

Example

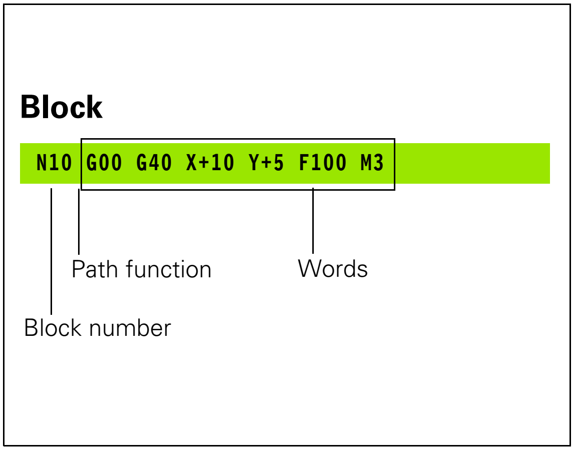
The workpiece drawing shows holes (**1** to **4**) whose dimensions are shown with respect to an absolute datum with the coordinates X=0 Y=0. Holes **5** to **7** are dimensioned with respect to a relative datum with the absolute coordinates X=450, Y=750. With the **DATUM SHIFT** cycle you can temporarily set the datum to the position X=450, Y=750, to be able to program holes **5** to **7** without further calculations.



3.2 Creating and Writing Programs

Organization of an NC program in DIN/ISO

A part program consists of a series of program blocks. The figure at right illustrates the elements of a block.



The TNC numbers the blocks of a part program automatically depending on machine parameter **blockIncrement** (105409). The machine parameter **blockIncrement** (105409) defines the block number increment.

The first block of a program is identified by `%`, the program name and the active unit of measure.

The subsequent blocks contain information on:

- The workpiece blank
- Tool calls
- Approaching a safe position
- Feed rates and spindle speeds, as well as
- Path contours, cycles and other functions

The last block of a program is identified by **`N99999999`** the program name and the active unit of measure.

After each tool call, HEIDENHAIN recommends always traversing to a safe position from which the TNC can position the tool for machining without causing a collision!

Define the blank: G30/G31

Immediately after initiating a new program, you define a cuboid workpiece blank. If you wish to define the blank at a later stage, press the SPEC FCT key, the PROGRAM DEFAULTS soft key, and then the BLK FORM soft key. This definition is needed for the TNC's graphic simulation feature. The sides of the workpiece blank lie parallel to the X, Y and Z axes and can be up to 100 000 mm long. The blank form is defined by two of its corner points:

- MIN point G30: the smallest X, Y and Z coordinates of the blank form, entered as absolute values
- MAX point G31: the largest X, Y and Z coordinates of the blank form, entered as absolute or incremental values

You only need to define the blank form if you wish to run a graphic test for the program!

Creating a new part program

You always enter a part program in the **Programming and Editing** mode of operation. An example of program initiation:

Select the **Programming and Editing** operating mode

PGM MGT

Press the PGM MGT key to call the file manager

Select the directory in which you wish to store the new program:

FILE NAME = ALT.I

ENT

Enter the new program name and confirm your entry with the ENT key

MM

To select the unit of measure, press the MM or INCH soft key. The TNC switches the screen layout and initiates the dialog for defining the **BLK FORM** (workpiece blank)

WORKING PLANE IN GRAPHIC: XY

Z

Enter spindle axis, e.g. Z

WORKPIECE BLANK DEF.: MINIMUM

ENT

Enter in sequence the X, Y and Z coordinates of the MIN point and confirm each of your entries with the ENT key

WORKPIECE BLANK DEF.: MAXIMUM

ENT

Enter in sequence the X, Y and Z coordinates of the MAX point and confirm each of your entries with the ENT key

Example: Display the BLK form in the NC program

`%NEW G71 *`	Program begin, name, unit of measure
`N10 G30 G17 X+0 Y+0 Z-40 *`	Spindle axis, MIN point coordinates
`N20 G31 X+100 Y+100 Z+0 *`	MAX point coordinates
`N99999999 %NEW G71 *`	Program end, name, unit of measure

The TNC automatically generates the first and last blocks of the program.

If you do not wish to define a blank form, cancel the dialog at **`Working plane in graphic: XY`** by pressing the DEL key.

The TNC can display the graphics only if the shortest side is at least 50 µm long and the longest side is no longer than 99 999.999 mm.

Programming tool movements in DIN/ISO format

Press the SPEC FCT key to program a block. Press the PROGRAM FUNCTIONS soft key, and then the DIN/ISO soft key. You can also use the gray contouring keys to get the corresponding G code.

If you enter DIN/ISO functions via a connected USB keyboard, make sure that capitalization is active.

Example of a positioning block

G 1 ENT Start block.

COORDINATES?

X 10 Enter the target coordinate for the X axis

Y 20 ENT Enter the target coordinate for the Y axis, and go to the next question with ENT

PATH OF THE CUTTER CENTER

G 40 Select tool movement without radius compensation: Confirm with the ENT key or

G41 G42 To move the tool to the left or to the right of the contour, select function G41 (to the left) or G42 (to the right) by soft key

FEED RATE F=?

100 ENT Enter a feed rate of 100 mm/min for this path contour; go to the next question with ENT

MISCELLANEOUS FUNCTION M?

3 ENT Enter the miscellaneous function **M3** "spindle ON." Pressing the ENT key terminates this dialog

The program-block window displays the following line:

```
N30 G01 G40 X+10 Y+5 F100 M3 *
```

Actual position capture

The TNC enables you to transfer the current tool position into the program, for example during

- Positioning-block programming
- Cycle programming

To transfer the correct position values, proceed as follows:

- ▶ Place the input box at the position in the block where you want to insert a position value

- ▶ Select the actual-position-capture function. In the soft-key row the TNC displays the axes whose positions can be transferred

- ▶ Select the axis. The TNC writes the current position of the selected axis into the active input box

In the working plane the TNC always captures the coordinates of the tool center, even though tool radius compensation is active.

In the tool axis the TNC always captures the coordinates of the tool tip and thus always takes the active tool length compensation into account.

The TNC keeps the soft-key row for axis selection active until you deactivate it by pressing the actual-position-capture key again. This behavior remains in effect even if you save the current block and open a new one with a path function key. If you select a block element in which you must choose an input alternative via soft key (e.g. for radius compensation), then the TNC also closes the soft-key row for axis selection.

The actual-position-capture function is not allowed if the tilted working plane function is active.

Editing a program

> You cannot edit a program while it is being run by the TNC in a machine operating mode.

While you are creating or editing a part program, you can select any desired line in the program or individual words in a block with the arrow keys or the soft keys:

Function	Soft key/Keys
Go to previous page	PAGE
Go to next page	PAGE
Go to beginning of program	BEGIN
Go to end of program	END
Change the position of the current block on the screen. Press this soft key to display additional program blocks that are programmed before the current block.	
Change the position of the current block on the screen. Press this soft key to display additional program blocks that are programmed after the current block.	
Move from one block to the next	
Select individual words in a block	
To select a certain block, press the GOTO key, enter the desired block number, and confirm with the ENT key. Or: Enter the block number step and press the N LINES soft key to jump over the entered number of lines upward or downward.	GOTO

Function	Soft key/Key
Set the selected word to zero	CE
Erase an incorrect number	CE
Clear a (non-blinking) error message	CE
Delete the selected word	NO ENT
Delete the selected block	DEL
Erase cycles and program sections	DEL
Insert the block that you last edited or deleted	INSERT LAST NC BLOCK

Inserting blocks at any desired location

- Select the block after which you want to insert a new block and initiate the dialog

Editing and inserting words

- Select a word in a block and overwrite it with the new one. The plain-language dialog is available while the word is highlighted
- To accept the change, press the END key

If you want to insert a word, press the horizontal arrow key repeatedly until the desired dialog appears. You can then enter the desired value.

Looking for the same words in different blocks

To use this function, set the AUTO DRAW soft key to OFF.

→	To select a word in a block, press the arrow keys repeatedly until the highlight is on the desired word
↓	Select a block with the arrow keys

The word that is highlighted in the new block is the same as the one you selected previously.

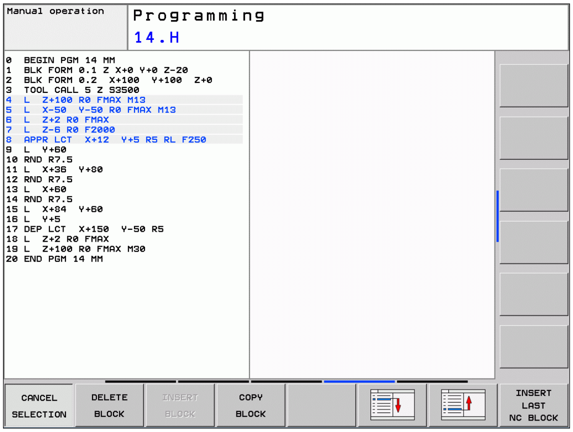
If you have started a search in a very long program, the TNC shows a progress display window. You then have the option of canceling the search via soft key.

Finding any text

- To select the search function, press the FIND soft key. The TNC displays the **`Find text:`** dialog prompt
- Enter the text that you wish to find
- To find the text, press the EXECUTE soft key

Marking, copying, deleting and inserting program sections

The TNC provides certain functions for copying program sections within an NC program or into another NC program—see the table below.



To copy a program section, proceed as follows:

- Select the soft-key row containing the marking functions
- Select the first (last) block of the section you wish to copy
- To mark the first (last) block, press the SELECT BLOCK soft key. The TNC then highlights the first character of the block and the CANCEL SELECTION soft key appears
- Move the highlight to the last (first) block of the program section you wish to copy or delete. The TNC shows the marked blocks in a different color. You can end the marking function at any time by pressing the CANCEL SELECTION soft key
- To copy the selected program section, press the COPY BLOCK soft key. To delete the selected section, press the DELETE BLOCK soft key. The TNC stores the selected block
- Using the arrow keys, select the block after which you wish to insert the copied (deleted) program section

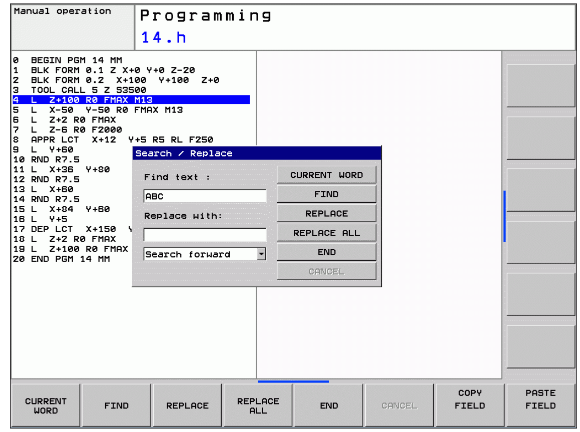
To insert the section into another program, select the corresponding program using the file manager and then mark the block after which you wish to insert the copied block.

- To insert the block, press the INSERT BLOCK soft key
- To end the marking function, press the CANCEL SELECTION soft key

Function	Soft key
Switch the marking function on	SELECT BLOCK
Switch the marking function off	CANCEL SELECTION
Delete the marked block	CUT OUT BLOCK
Insert the block that is stored in the buffer memory	INSERT BLOCK
Copy the marked block	COPY BLOCK

The TNC search function

With the search function of the TNC, you can search for any text within a program and replace it by a new text, if required.



Searching for texts

- If required, select the block containing the word you wish to find

FIND

- Select the search function. The TNC superimposes the search window and displays the available search functions in the soft-key row (see table of search functions)

X +40

- Enter the text to be searched for. Please note that the search is case-sensitive

FIND

- Start the search process: The TNC moves to the next block containing the text you are searching for

FIND

- Repeat the search process: The TNC moves to the next block containing the text you are searching for

END

- End the search function

Find/Replace any text

> The find/replace function is not possible if
>
> - a program is protected
> - the program is currently being run by the TNC
>
> When using the REPLACE ALL function, ensure that you do not accidentally replace text that you do not want to change. Once replaced, such text cannot be restored.

- If required, select the block containing the word you wish to find

FIND

- Select the Search function: The TNC superimposes the search window and displays the available search functions in the soft-key row

X

- Enter the text to be searched for. Please note that the search is case-sensitive. Then confirm with the ENT key

Z

- Enter the text to be inserted. Please note that the entry is case-sensitive

FIND

- Start the search process: The TNC moves to the next occurrence of the text you are searching for

REPLACE

- To replace the text and then move to the next occurrence of the text, press the REPLACE soft key. To replace all text occurrences, press the REPLACE ALL soft key. To skip the text and move to its next occurrence press the FIND soft key

FIND

- End the search function

3.3 File Management: Fundamentals

Files

Files in the TNC	Type
Programs	
In HEIDENHAIN format	.H
In DIN/ISO format	.I
Tables for	
Tools	.T
Tool changers	.TCH
Pallets	.P
Datums	.D
Points	.PNT
Presets	.PR
Touch probes	.TP
Backup files	.BAK
Texts as	
ASCII files	.A
Log files	.TXT
Help files	.CHM

When you write a part program on the TNC, you must first enter a file name. The TNC saves the program to the hard disk as a file with the same name. The TNC can also save texts and tables as files.

The TNC provides a special file management window in which you can easily find and manage your files. Here you can call, copy, rename and erase files.

With the TNC you can manage and save files up to a total size of 300 MB.

Depending on the setting, the TNC generates a backup file (*.bak) after editing and saving of NC programs. This can reduce the memory space available to you.

File names

When you store programs, tables and texts as files, the TNC adds an extension to the file name, separated by a point. This extension indicates the file type.

PROG20	.H
File name	File type

File names should not exceed 25 characters, otherwise the TNC cannot display the entire file name. The following characters are not permitted in file names:

! " ' ( ) * + / ; < = > ? [ ] ^ ` { | } ~

Enter the file name using the screen keyboard (see "Screen Keyboard" on page 110).

The space (HEX 20) and delete (HEX 7F) characters are not permitted in file names, either.

The maximum limit for the path and file name together is 256 characters (see "Paths" on page 92).

Data backup

We recommend saving newly written programs and files on a PC at regular intervals.

The TNCremoNT data transmission freeware from HEIDENHAIN is a simple and convenient method for backing up data stored on the TNC.

You additionally need a data medium on which all machine-specific data, such as the PLC program, machine parameters, etc., are stored. Ask your machine manufacturer for assistance, if necessary.

Take the time occasionally to delete any unneeded files so that the TNC always has enough memory space for system files (such as the tool table).

3.4 Working with the File Manager

Directories

To ensure that you can easily find your files, we recommend that you organize your hard disk into directories. You can divide a directory into further directories, which are called subdirectories. With the –/+ key or ENT you can show or hide the subdirectories.

Paths

A path indicates the drive and all directories and subdirectories under which a file is saved. The individual names are separated by a backslash "\".

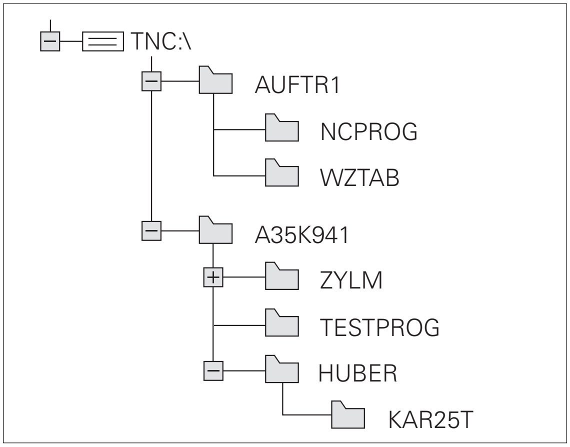
The path, including all drive characters, directory and the file name, including the extension, cannot exceed 256 characters!

Example

The directory AUFTR1 was created on the **TNC:** drive. Then, in the **AUFTR1** directory, the directory NCPROG was created and the part program PROG1.H was copied into it. The part program now has the following path:

TNC:\AUFTR1\NCPROG\PROG1.H

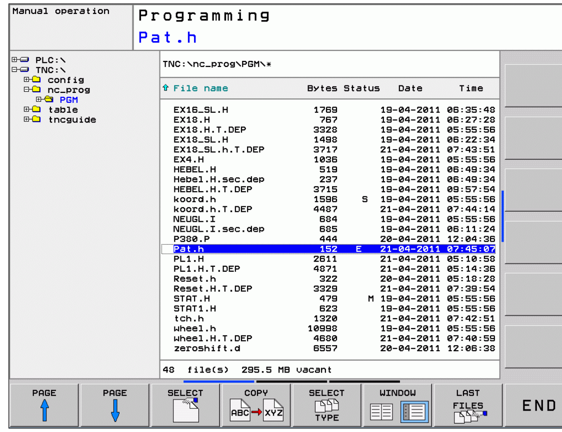
The chart at right illustrates an example of a directory display with different paths.



Overview: Functions of the file manager

Function	Soft key	Page
Copy a file	COPY ABC → XYZ	Page 98
Display a specific file type	SELECT TYPE	Page 95
Create new file	NEW FILE	Page 97
Display the last 10 files that were selected	LAST FILES	Page 99
Delete a file or directory	DELETE	Page 99
Tag a file	TAG	Page 101
Rename a file	RENAME ABC = XYZ	Page 102
Protect a file against editing and erasure	PROTECT	Page 103
Cancel file protection	UNPROTECT	Page 103
Import tool table	IMPORT TABLE	Page 140
Manage network drives	NET	Page 106
Select the editor	SELECT EDITOR	Page 103
Sort files by properties	SORT	Page 102
Copy a directory	COPY DIR	Page 98
Delete directory with all its subdirectories	DELETE ALL	
Display all the directories of a particular drive	UPDATE TREE	
Rename a directory	RENAME ABC = XYZ	
Create a new directory	NEW DIRECTORY	

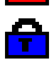
Calling the file manager



PGM MGT	Press the PGM MGT key: The TNC displays the file management window (see figure for default setting. If the TNC displays a different screen layout, press the WINDOW soft key.)

The narrow window on the left shows the available drives and directories. Drives designate devices with which data are stored or transferred. One drive is the hard disk of the TNC. Other drives are the interfaces (RS232, Ethernet), which can be used, for example, for connecting a personal computer. A directory is always identified by a folder symbol to the left and the directory name to the right. Subdirectories are shown to the right of and below their parent directories. A triangle in front of the folder symbol indicates that there are further subdirectories, which can be shown with the –/+ or ENT keys.

The wide window on the right shows you all files that are stored in the selected directory. Each file is shown with additional information, illustrated in the table below.

Display	Meaning
`File name`	Name with max. 25 characters
`Type`	File type
`Bytes`	File size in bytes
`Status`	File properties:
E	Program is selected in the Programming mode of operation.
S	Program is selected in the Test Run mode of operation.
M	Program is selected in a Program Run mode of operation.
	File is protected against erasing and editing
	File is protected against erasing and edition, because it is being run
`Date`	Date that the file was last edited
`Time`	Time that the file was last edited

Selecting drives, directories and files

PGM MGT	Call the file manager

Use the arrow keys or the soft keys to move the highlight to the desired position on the screen:

← →	Moves the highlight from the left to the right window, and vice versa
↓ ↑	Moves the highlight up and down within a window
PAGE ↑ PAGE ↓	Moves the highlight one page up or down within a window

Step 1: Select drive

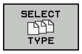
Move the highlight to the desired drive in the left window:

SELECT	To select a drive, press the SELECT soft key, or
ENT	Press the ENT key

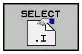
Step 2: Select a directory

Move the highlight to the desired directory in the left-hand window—the right-hand window automatically shows all files stored in the highlighted directory

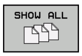
Step 3: Select a file



Press the SELECT TYPE soft key

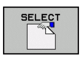


Press the soft key for the desired file type, or



Press the SHOW ALL soft key to display all files, or

Move the highlight to the desired file in the right window:



Press the SELECT soft key, or

Press the ENT key

The TNC opens the selected file in the operating mode from which you called the file manager

Creating a new directory

Move the highlight in the left window to the directory in which you want to create a subdirectory

NEW ENT — Enter the new file name, and confirm with ENT

CREATE \NEW DIRECTORY?

YES — Press the YES soft key to confirm, or

NO — abort with the NO soft key

Creating a new file

Select the directory in which you wish to create the new file

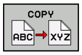
NEW ENT — Enter the new file name with the file extension, and confirm with ENT

NEW FILE — Open the dialog box for creating a new file

NEW ENT — Enter the new file name with the file extension, and confirm with ENT

Copying a single file

- Move the highlight to the file you wish to copy



- Press the COPY soft key to select the copy function. The TNC opens a pop-up window

- Enter the name of the destination file and confirm your entry with the ENT key or OK soft key: the TNC copies the file to the active directory or to the selected destination directory. The original file is retained, or:

Copying files into another directory

- Select a screen layout with two equally sized windows
- To display directories in both windows, press the PATH soft key

In the right window

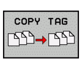
- Move the highlight to the directory into which you wish to copy the files, and display the files in this directory with the ENT key

In the left window

- Select the directory with the files that you wish to copy and press ENT to display them

- Call the file tagging functions

- Move the highlight to the file you want to copy and tag it. You can tag several files in this way, if desired



- Copy the tagged files into the target directory

Additional tagging functions: see "Marking files", page 101.

If you have tagged files in both the left and right windows, the TNC copies from the directory in which the highlight is located.

Copying a directory

- Move the highlight in the right window onto the directory you want to copy
- Press the COPY soft key: the TNC opens the window for selecting the target directory
- Select the target directory and confirm with ENT or the OK soft key. The TNC copies the selected directory and all its subdirectories to the selected target directory

Choosing one of the last files selected

PGM MGT — Call the file manager

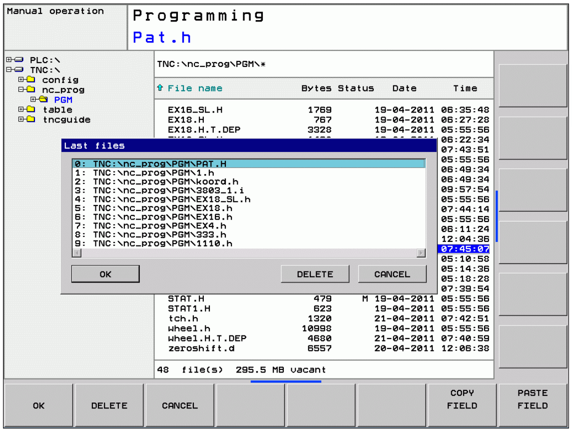
LAST FILES — Display the last 10 files selected: Press the LAST FILES soft key

Use the arrow keys to move the highlight to the file you wish to select:

Moves the highlight up and down within a window

OK — To select the file, press the OK soft key, or

ENT — Press the ENT key



Deleting a file

> Once you delete files they cannot be restored!

- Move the highlight to the file you want to delete.

DELETE

- To select the erasing function, press the DELETE soft key. The TNC asks whether you really want to delete the file
- To confirm, press the OK soft key, or
- To cancel deletion, press the CANCEL soft key

Deleting a directory

Once you delete directories they cannot be restored!

- Move the highlight to the directory you want to delete

- To select the erasing function, press the DELETE soft key. The TNC inquires whether you really intend to delete the directory and all its subdirectories and files
- To confirm, press the OK soft key, or
- to cancel deletion, press the CANCEL soft key

Marking files

Marking function	Soft key
Mark a single file	TAG FILE
Mark all files in the directory	TAG ALL FILES
Unmark a single file	UNTAG FILE
Unmark all files	UNTAG ALL FILES
Copy all marked files	COPY TAG

Some functions, such as copying or erasing files, can not only be used for individual files, but also for several files at once. To mark several files, proceed as follows:

Move the highlight to the first file

TAG — To display the marking functions, press the TAG soft key

TAG FILE — Mark a file by pressing the TAG FILE soft key

↑ ↓ — Move the highlight to the next file you wish to mark: Only works via soft keys. Do not use the arrow keys!

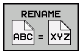
TAG FILE — To mark further files, press the TAG FILE soft key, etc.

COPY TAG — To copy the marked files, press the COPY TAG soft key, or

END DELETE — Delete the marked files by pressing END to end the marking function, and then the DELETE soft key to delete the marked files

Renaming a file

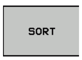
- Move the highlight to the file you wish to rename



- Select the renaming function
- Enter the new file name; the file type cannot be changed
- To rename: Press the OK soft key or the ENT key

File sorting

- Select the folder in which you wish to sort the files



- Select the SORT soft key
- Select the soft key with the corresponding display criterion

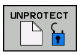
Additional functions

Protecting a file / Canceling file protection

- Move the highlight to the file you want to protect

- To select the additional functions, press the MORE FUNCTIONS soft key

- To activate file protection, press the PROTECT soft key. The file now has status P



- To cancel file protection, press the UNPROTECT soft key

Select the editor

- Move the highlight in the right window onto the file you want to open

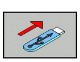
- To select the additional functions, press the MORE FUNCTIONS soft key

- To select the editor with which to open the selected file, press the SELECT EDITOR soft key
- Mark the desired editor
- Press the OK soft key to open the file

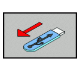
Connecting/removing a USB device

- Move the highlight to the left window

- To select the additional functions, press the MORE FUNCTIONS soft key
- Shift the soft-key row



- Search for a USB device
- In order to remove the USB device, move the cursor to the USB device



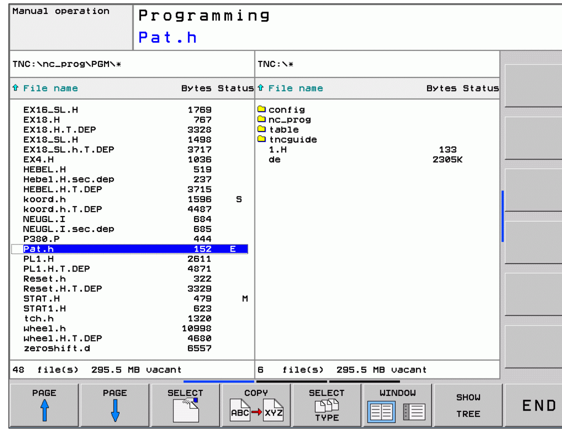
- Remove the USB device

For more information: See "USB devices on the TNC" on page 107.

Data transfer to or from an external data medium

Before you can transfer data to an external data medium, you must set up the data interface (see "Setting the Data Interfaces" on page 398).

Depending on the data transfer software you use, problems can occur occasionally when you transmit data over a serial interface. They can be overcome by repeating the transmission.



PGM MGT — Call the file manager

WINDOW — Select the screen layout for data transfer: press the WINDOW soft key. In the left half of the screen the TNC shows all files in the current directory. In the right half of the screen it shows all files saved in the root directory (TNC:\ ).

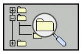
Use the arrow keys to highlight the file(s) that you want to transfer:

↑ ↓ — Moves the highlight up and down within a window

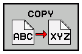
← → — Moves the highlight from the left to the right window, and vice versa

If you wish to copy from the TNC to the external data medium, move the highlight in the left window to the file to be transferred.

If you wish to copy from the external data medium to the TNC, move the highlight in the right window to the file to be transferred.



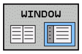
To select another drive or directory: press the soft key for choosing the directory. The TNC opens a pop-up window. Select the desired directory in the pop-up window by using the arrow keys and the ENT key



Transfer a single file: Press the COPY soft key, or

to transfer several files, press the TAG soft key (in the second soft-key row, see "Marking files", page 101)

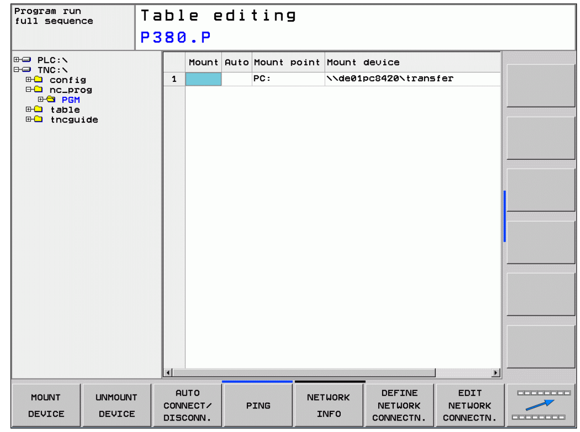
Confirm with the OK soft key or with the ENT key. A status window appears on the TNC, informing about the copying progress, or



To end data transfer, move the highlight into the left window and then press the WINDOW soft key. The standard file manager window is displayed again

To select another directory in the split-screen display, press the SHOW TREE soft key. If you press the SHOW FILES soft key, the TNC shows the content of the selected directory!

The TNC in a network



> To connect the Ethernet card to your network, see "Ethernet Interface", page 403.
>
> The TNC logs error messages during network operation see "Ethernet Interface", page 403.

If the TNC is connected to a network, the directory window displays additional drives (see figure). All the functions described above (selecting a drive, copying files, etc.) also apply to network drives, provided that you have been granted the corresponding rights.

Connecting and disconnecting a network drive

PGM MGT

- To select the program management: Press the PGM MGT key. If necessary, press the WINDOW soft key to set up the screen as it is shown at the upper right

NET

- To manage the network drives: Press the NETWORK soft key (second soft-key row). In the right-hand window the TNC shows the network drives available for access. With the soft keys described below you can define the connection for each drive

Function	Soft key
Establish the network connection. If the connection is active, the TNC marks the **Mnt** column	MOUNT DEVICE
Delete the network connection	UNMOUNT DEVICE
Automatically establish network connection whenever the TNC is switched on. The TNC marks the **Auto** column if the connection is established automatically	AUTO MOUNT
Use the PING function to test your network connection	PING
If you press the NETWORK INFO soft key, the TNC displays the current network settings	NETWORK INFO

USB devices on the TNC

Backing up data from or loading onto the TNC is especially easy with USB devices. The TNC supports the following USB block devices:

- Floppy disk drives with FAT/VFAT file system
- Memory sticks with the FAT/VFAT file system
- Hard disks with the FAT/VFAT file system
- CD-ROM drives with the Joliet (ISO 9660) file system

The TNC automatically detects these types of USB devices when connected. The TNC does not support USB devices with other file systems (such as NTFS). The TNC displays the **`USB: TNC does not support device`** error message when such a device is connected.

> The TNC also displays the **`USB: TNC does not support device`** error message if you connect a USB hub. In this case, simply acknowledge the message with the CE key.
>
> In theory, you should be able to connect all USB devices with the file systems mentioned above to the TNC. It may happen that a USB device is not identified correctly by the control. In such cases, use another USB device.

The USB devices appear as separate drives in the directory tree, so you can use the file-management functions described in the earlier chapters correspondingly.

To remove a USB device, proceed as follows:

PGM MGT ▶ Press the PGM MGT soft key to call the file manager

← ▶ Select the left window with the arrow key

↓ ▶ Use the arrow keys to select the USB device to be removed

▷ ▶ Scroll through the soft-key row

NET ▶ Select additional functions.


▶ Select the function for removing USB devices. The TNC removes the USB device from the directory tree

END □ ▶ Exit the file manager

In order to re-establish a connection with a USB device that has been removed, press the following soft key:

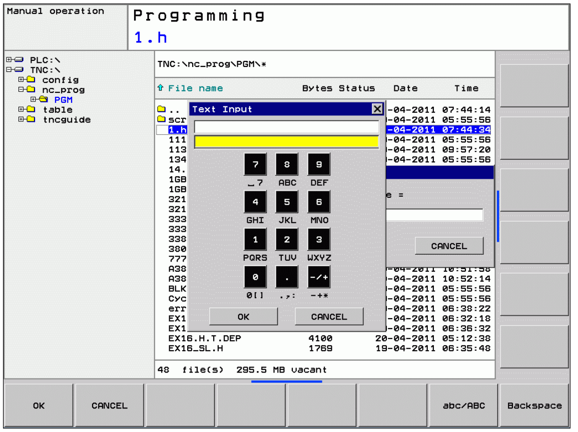

▶ Select the function for reconnection of USB devices

4

Programming: Programming Aids

4.1 Screen Keyboard

You can enter letters and special characters with the screen keypad or (if available) with a PC keyboard connected over the USB port.



Enter the text with the screen keyboard

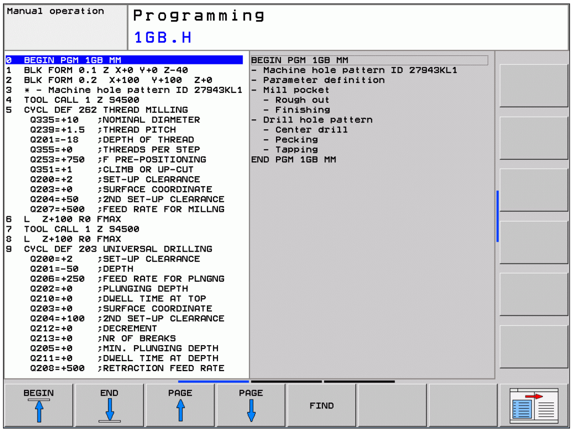
- Press the GOTO key if you want to enter a text, for example a program name or directory name, using the screen keyboard
- The TNC opens a window in which the numeric entry field of the TNC is displayed with the corresponding letters assigned
- You can move the cursor to the desired character by repeatedly pressing the respective key
- Wait until the selected character is transferred to the entry field before you enter the next character
- Use the OK soft key to load the text into the open dialog field

Use the **abc/ABC** soft key to select upper or lower case. If your machine tool builder has defined additional special characters, you can call them with the SPECIAL CHARACTER soft key and insert them. To delete individual characters, use the BACKSPACE soft key.

4.2 Adding Comments

Application

You can add comments to a part program to explain program steps or make general notes.



> Enter the file name using the screen keyboard (see "Screen Keyboard" on page 110).
>
> If the TNC cannot show the entire comment on the screen, the **>>** sign is displayed.
>
> The last character in a comment block must not have any tilde (**~**).

Entering a comment in a separate block

- Select the block after which the comment is to be inserted.
- Press the SPEC FCT key to select the special functions.
- To select the program functions, press the PROGRAM FUNCTIONS soft key.
- Shift soft-key row to the left
- Press the INSERT COMMENT soft key.
- Enter your comment using the screen keyboard (see "Screen Keyboard" on page 110) and conclude the block by pressing the END key.

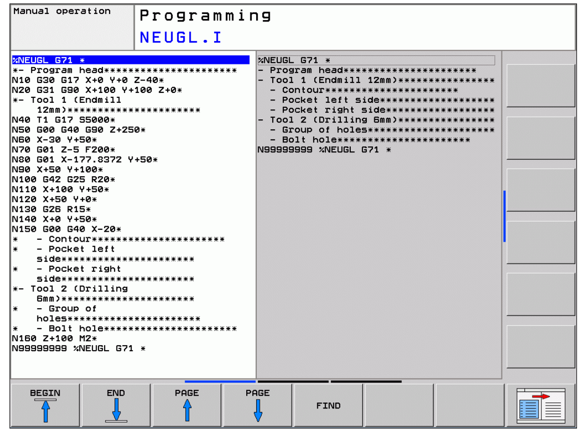
> If you have connected a PC keyboard to the USB interface, you can insert a comment block by simply pressing the **;** key on the PC keyboard.

Functions for editing of the comment

Function	Soft key
Jump to beginning of comment.	BEGIN
Jump to end of comment.	END
Jump to the beginning of a word. Words must be separated by a space.	MOVE WORD ←
Jump to the end of a word. Words must be separated by a space.	MOVE WORD →
Switch between insert mode and overwrite mode.	INSERT OVERWRITE

4.3 Structuring Programs

Definition and applications



This TNC function enables you to comment part programs in structuring blocks. Structuring blocks are short texts with up to 37 characters and are used as comments or headlines for the subsequent program lines.

With the aid of appropriate structuring blocks, you can organize long and complex programs in a clear and comprehensible manner.

This function is particularly convenient if you want to change the program later. Structuring blocks can be inserted into the part program at any point. They can also be displayed in a separate window, and edited or added to, as desired.

The inserted structure items are managed by the TNC in a separate file (extension: .SEC.DEP). This speeds navigation in the program structure window.

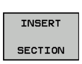
Displaying the program structure window / Changing the active window

- To display the program structure window, select the screen display PROGRAM+SECTS

- To change the active window, press the "Change window" soft key

Inserting a structuring block in the (left) program window

- Select the block after which the structuring block is to be inserted



- Press the INSERT SECTION soft key or the * key on the ASCII keyboard
- Enter the structuring text with the alphabetic keyboard

- If necessary, change the structure depth with the soft key

Selecting blocks in the program structure window

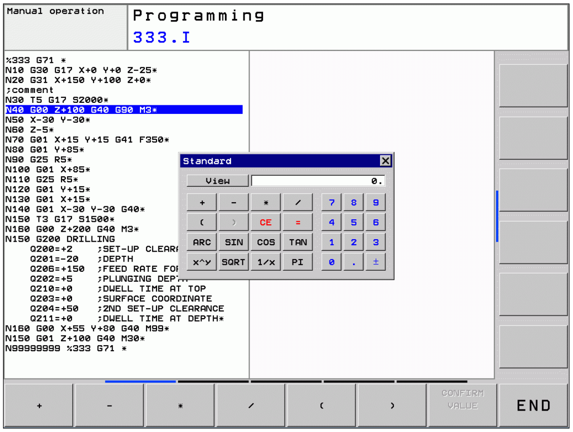
If you are scrolling through the program structure window block by block, the TNC at the same time automatically moves the corresponding NC blocks in the program window. This way you can quickly skip large program sections.

4.4 Integrated Pocket Calculator

Operation

The TNC features an integrated pocket calculator with the basic mathematical functions.

- Use the CALC key to show and hide the on-line pocket calculator.
- The calculator is operated with short commands through the alphabetic keyboard. The commands are shown in a special color in the calculator window:



Mathematical function	Command (key)
Addition	+
Subtraction	–
Multiplication	*
Division	/
Calculations in parentheses	( )
Arc cosine	ARC
Sine	SIN
Cosine	COS
Tangent	TAN
Powers of values	X^Y
Square root	SQRT
Inversion	1/x
pi (3.14159265359)	PI
Add value to buffer memory	M+
Save the value to buffer memory	MS
Recall from buffer memory	MR
Delete buffer memory contents	MC
Natural logarithm	LN
Logarithm	LOG
Exponential function	e^x
Check the algebraic sign	SGN
Form the absolute value	ABS

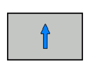
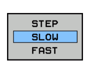
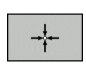
Mathematical function	Command (key)
Truncate decimal places	INT
Truncate integers	FRAC
Modulus operator	MOD
Select view	View
Delete value	CE
Unit of measure	MM or INCH
Display mode for angle values	DEG (degree) or RAD (radian measure)
Display mode of the numerical value	DEC (decimal) or HEX (hexadecimal)

To transfer the calculated value into the program

- Use the arrow keys to select the word into which the calculated value is to be transferred
- Superimpose the on-line calculator by pressing the CALC key and perform the desired calculation
- Press the actual-position-capture key for the TNC to superimpose a soft-key row
- Press the CALC soft key for the TNC to transfer the value into the active input box and to close the calculator.

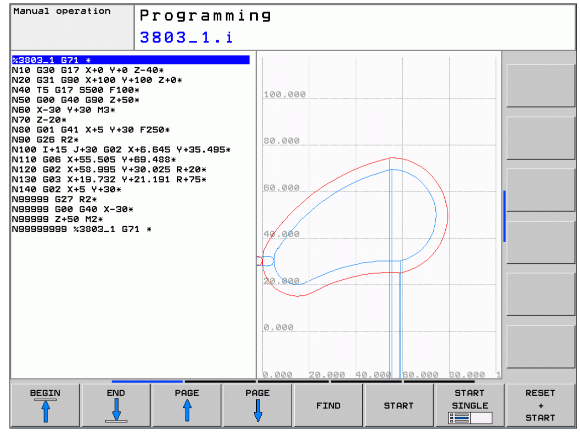
Adjusting the position of the calculator

Press the ADDITIONAL FUNCTIONS soft key to get to the settings for shifting the calculator:

Function	Soft key
Move calculator in the direction of the arrow	
Adjust the increment for movement	
Position the calculator in the center	

4.5 Programming Graphics

Generating / not generating graphics during programming



While you are writing the part program, you can have the TNC generate a 2-D pencil-trace graphic of the programmed contour.

▶ To switch the screen layout to displaying program blocks to the left and graphics to the right, press the SPLIT SCREEN key and PROGRAM + GRAPHICS soft key

AUTO DRAW OFF ON

▶ Set the AUTO DRAW soft key to ON. While you are entering the program lines, the TNC generates each path contour you program in the graphics window in the right screen half

If you do not wish to have the TNC generate graphics during programming, set the AUTO DRAW soft key to OFF.

Even when AUTO DRAW ON is active, graphics are not generated for program section repeats.

Generating a graphic for an existing program

▶ Use the arrow keys to select the block up to which you want the graphic to be generated, or press GOTO and enter the desired block number

RESET + START

▶ To generate graphics, press the RESET + START soft key

Additional functions:

Function	Soft key
Generate a complete graphic	RESET + START
Generate programming graphic blockwise	START SINGLE
Generate a complete graphic or complete it after RESET + START	START
Stop the programming graphics. This soft key only appears while the TNC is generating the interactive graphics	STOP

Block number display ON/OFF

SHOW OMIT BLOCK NR.

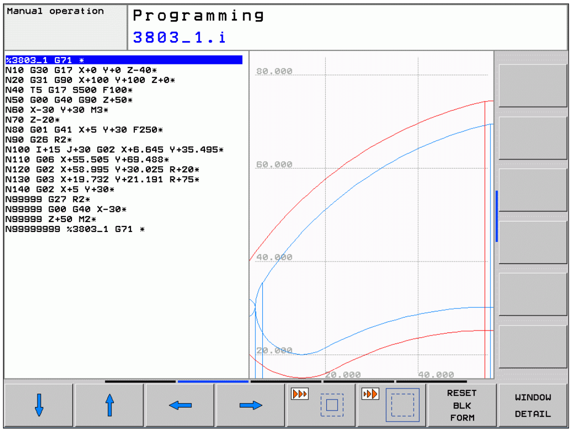
- Shift the soft-key row: see figure
- To show block numbers: Set the SHOW OMIT BLOCK NR. soft key to SHOW
- To omit block numbers: Set the SHOW OMIT BLOCK NR. soft key to OMIT

Erasing the graphic

CLEAR GRAPHICS

- Shift the soft-key row: see figure
- Erase graphic: Press CLEAR GRAPHICS soft key

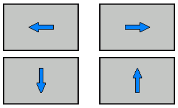
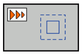
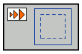
Magnifying or reducing a detail



You can select the graphics display by selecting a detail with the frame overlay. You can now magnify or reduce the selected detail.

- Select the soft-key row for detail magnification/reduction (second row, see figure)

The following functions are available:

Function	Soft key
Show and move the frame overlay. Press and hold the desired soft key to move the frame overlay	
Reduce the frame overlay—press and hold the soft key to reduce the detail	
Enlarge the frame overlay—press and hold the soft key to magnify the detail	

WINDOW DETAIL

- Confirm the selected area with the WINDOW DETAIL soft key

With the RESET WORKPIECE BLANK soft key, you can restore the original section.

4.6 Error Messages

Display of errors

The TNC generates error messages when it detects problems such as:

- Incorrect data input
- Logical errors in the program
- Contour elements that are impossible to machine
- Incorrect use of touch probes

When an error occurs, it is displayed in red type in the header. Long and multi-line error messages are displayed in abbreviated form. If an error occurs in the background mode, the word "Error" is displayed in red type. Complete information on all pending errors is shown in the error window.

If a rare "processor check error" should occur, the TNC automatically opens the error window. You cannot remove such an error. Shut down the system and restart the TNC.

The error message is displayed in the header until it is cleared or replaced by a higher-priority error.

An error message that contains a program block number was caused by an error in the indicated block or in the preceding block.

Open the error window

ERR

▶ Press the ERR key. The TNC opens the error window and displays all accumulated error messages.

Close the error window

END

▶ Press the END soft key—or

ERR

▶ Press the ERR key. The TNC closes the error window.

Detailed error messages

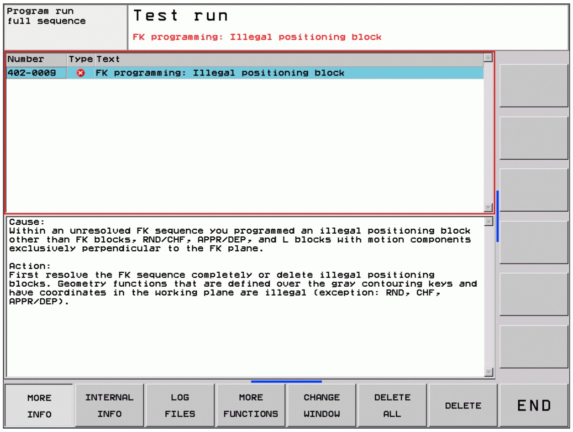
The TNC displays possible causes of the error and suggestions for solving the problem:

▶ Open the error window

MORE INFO

▶ Information on the error cause and corrective action: Position the highlight on the error message and press the MORE INFO soft key. The TNC opens a window with information on the error cause and corrective action.

▶ Leave Info: Press the MORE INFO soft key again.



INTERNAL INFO soft key

The INTERNAL INFO soft key supplies information on the error message. This information is only required if servicing is needed.

▶ Open the error window

INTERNAL INFO

▶ Detailed information about the error message: Position the highlight on the error message and press the INTERNAL INFO soft key. The TNC opens a window with internal information about the error

▶ To leave Details, press the INTERNAL INFO soft key again.

Clearing errors

Clearing errors outside of the error window:

CE

- To clear the error/message in the header: Press the CE button.

In some operating modes (such as the Editing mode), the CE button cannot be used to clear the error, since the button is reserved for other functions.

Clearing more than one error:

- Open the error window

DELETE

- Clear individual errors: Position the highlight on the error message and press the DELETE soft key.

DELETE ALL

- Clear all errors: Press the DELETE ALL soft key.

If the cause of the error has not been removed, the error message cannot be deleted. In this case, the error message remains in the window.

Error log

The TNC stores errors and important events (e.g. system startup) in an error log. The capacity of the error log is limited. If the log is full, the TNC uses a second file. If this is also full, the first error log is deleted and written to again, and so on. To view the error history, switch between CURRENT FILE and PREVIOUS FILE.

- Open the error window

LOG FILES

- Press the LOG FILES soft key.

ERROR LOG

- To open the error log, press the ERROR LOG FILE soft key.

PREVIOUS FILE

- If you need the previous log file, press the PREVIOUS FILE soft key.

CURRENT FILE

- If you need the current log file, press the CURRENT FILE soft key.

The oldest entry is at the beginning of the error log file, and the most recent entry is at the end.

Keystroke log

The TNC stores keystrokes and important events (e.g. system startup) in a keystroke log. The capacity of the keystroke log is limited. If the keystroke log is full, the control switches to a second keystroke log. If this second file becomes full, the first keystroke log is cleared and written to again, and so on. To view the keystroke history, switch between CURRENT FILE and PREVIOUS FILE.

LOG FILES
- Press the LOG FILES soft key.

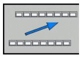
KEYSTROKE LOG
- To open the keystroke log file, press the KEYSTROKE LOG FILE soft key.

PREVIOUS FILE
- If you need the previous log file, press the PREVIOUS FILE soft key.

CURRENT FILE
- If you need the current log file, press the CURRENT FILE soft key.

The TNC saves each key pressed during operation in a keystroke log. The oldest entry is at the beginning, and the most recent entry is at the end of the file.

Overview of the buttons and soft keys for viewing the log files:

Function	Soft key/Keys
Go to beginning of log file	BEGIN
Go to end of log file	END
Current log file	CURRENT FILE
Previous log file	PREVIOUS FILE
Up/down one line	 
Return to main menu	

Informational texts

After a faulty operation, such as pressing a key without function or entering a value outside of the valid range, the TNC displays a (green) text in the header, informing you that the operation was not correct. The TNC clears this note upon the next valid input.

Saving service files

If necessary, you can save the “Current status of the TNC,” and make it available to a service technician for evaluation. A group of service files is saved (error and keystroke log files, as well as other files that contain information about the current status of the machine and the machining).

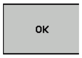
If you repeat the "Save service files" function with the same file name, the previously saved group of service data files is overwritten. To avoid this, use another file name when you repeat the function.

Saving service files:

- Open the error window

- Press the LOG FILES soft key.

- Press the SAVE SERVICE FILES soft key: The TNC opens a pop-up window in which you can enter a name for the service file

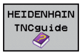


- To save service files, press the OK soft key.

Calling the TNCguide help system

You can call the TNC’s help system via soft key. Immediately the help system shows you the same error explanation that you receive by pressing the HELP soft key.

> If your machine manufacturer also provides a help system, the TNC shows an additional MACHINE MANUFACTURER soft key with which you can call this separate help system. There you will find further, more detailed information on the error message concerned.

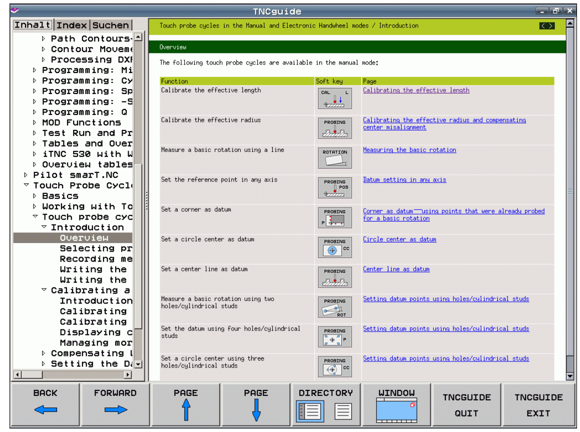


- Call the help for HEIDENHAIN error messages

- Call the help for HEIDENHAIN error messages, if available.

4.7 Context-Sensitive Help System

Application



Before you can use the TNCguide, you need to download the help files from the HEIDENHAIN home page (see "Downloading current help files" on page 128).

The **TNCguide** context-sensitive help system includes the user documentation in HTML format. The TNCguide is called with the HELP key, and the TNC often immediately displays the information specific to the condition from which the help was called (context-sensitive call). Even if you are editing an NC block and press the HELP key, you are usually brought to the exact place in the documentation that describes the corresponding function.

The TNC always tries to start the TNCguide in the language that you have selected as the conversational language on your TNC. If the files with this language are not yet available on your TNC, it automatically opens the English version.

The following user documentation is available in the TNCguide:

- Conversational Programming User's Manual **(BHBKlartext.chm)**
- DIN/ISO User's Manual **(BHBIso.chm)**
- User's Manual for Cycle Programming (**BHBtchprobe.chm**)
- List of All Error Messages **(errors.chm)**

In addition, the **main.chm** "book" file is available, with the contents of all existing .chm files.

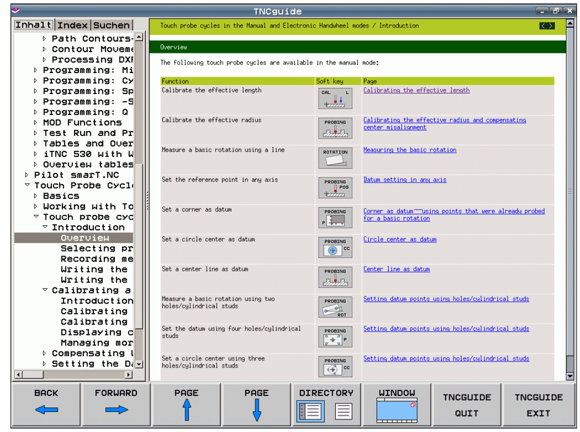
As an option, your machine tool builder can embed machine-specific documentation in the **TNCguide.** These documents then appear as a separate book in the **main.chm** file.

Working with the TNCguide

Calling the TNCguide

There are several ways to start the TNCguide:

- Press the HELP key if the TNC is not already showing an error message
- Click the help symbol at the lower right of the screen beforehand, then click the appropriate soft keys
- Use the file manager to open a help file (.chm file). The TNC can open any .chm file, even if it is not saved on the TNC's hard disk



> If one or more error messages are waiting for your attention, the TNC shows the help directly associated with the error messages. To start the **TNCguide,** you first have to acknowledge all error messages.
>
> When the help system is called on the programming station, the TNC starts the internally defined standard browser (usually the Internet Explorer), or otherwise a browser adapted by HEIDENHAIN.

For many soft keys there is a context-sensitive call through which you can go directly to the description of the soft key's function. This functionality requires using a mouse. Proceed as follows:

- Select the soft-key row containing the desired soft key
- Click with the mouse on the help symbol that the TNC displays just above the soft-key row: The mouse pointer turns into a question mark
- Move the question mark to the soft key for which you want an explanation, and click: The TNC opens the TNCguide. If no specific part of the help is assigned to the selected soft key, the TNC opens the book file **main.chm,** in which you can use the search function or the navigation to find the desired explanation manually

Even if you are editing an NC block, context-sensitive help is available:

- Select any NC block
- Use the arrow keys to move the cursor to the block
- Press the HELP key: The TNC starts the help system and shows a description for the active function (does not apply to miscellaneous functions or cycles that were integrated by your machine tool builder)

Navigating in the TNCguide

It’s easiest to use the mouse to navigate in the TNCguide. A table of contents appears on the left side of the screen. By clicking the rightward pointing triangle you open subordinate sections, and by clicking the respective entry you open the individual pages. It is operated in the same manner as the Windows Explorer.

Linked text positions (cross references) are shown underlined and in blue. Clicking the link opens the associated page.

Of course you can also operate the TNCguide through keys and soft keys. The following table contains an overview of the corresponding key functions.

> The key functions described below are only available on the control hardware, and not on the programming station.

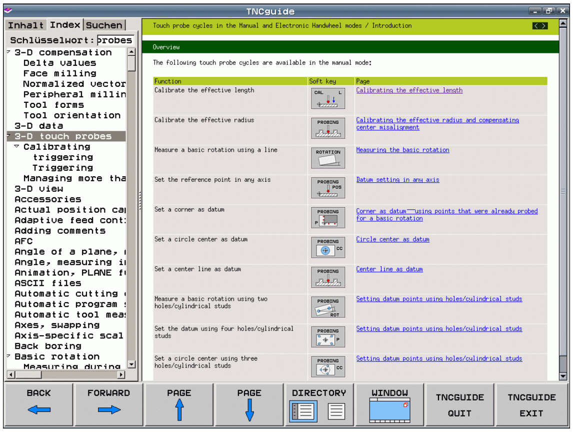
Function	Soft key
■ If the table of contents at left is active: Select the entry above it or below it ■ If the text window at right is active: Move the page downward or upward if texts or graphics are not shown completely	↓ ↑
■ If the table of contents at left is active: Open a branch of the table of contents. If the branch is at its end, jump into the window at right ■ If the text window at right is active: No function	→
■ If the table of contents at left is active: Close a branch of the table of contents ■ If the text window at right is active: No function	←
■ If the table of contents at left is active: Use the cursor key to show the selected page ■ If the text window at right is active: If the cursor is on a link, jump to the linked page	ENT
■ If the table of contents at left is active: Switch the tab between the display of the table of contents, display of the subject index, and the full-text search function and switching to the screen half at right ■ If the text window at right is active: Jump back to the window at left	
■ If the table of contents at left is active: Select the entry above it or below it ■ If the text window at right is active: Jump to the next link	 

Function	Soft key
Select the page last shown	BACK
Page forward if you have used the “select page last shown” function	FORWARD
Move up by one page	PAGE
Move down by one page	PAGE
Display or hide table of contents	DIRECTORY
Switch between full-screen display and reduced display. With the reduced display you can see some of the rest of the TNC window	WINDOW
The focus is switched internally to the TNC application so that you can operate the control when the TNCguide is open. If the full screen is active, the TNC reduces the window size automatically before the change of focus	TNCGUIDE QUIT
Close the TNCguide	TNCGUIDE EXIT

Subject index

The most important subjects in the Manual are listed in the subject index (**Index** tab). You can select them directly by mouse or with the cursor keys.

The left side is active.



- Select the **Index** tab
- Activate the **Keyword** input field
- Enter the word for the desired subject and the TNC synchronizes the index and creates a list in which you can find the subject more easily, or
- Use the arrow key to highlight the desired keyword
- Use the ENT key to call the information on the selected keyword

> You can enter the search word only with a keyboard connected via USB.

Full-text search

In the **Find** tab you can search the entire TNCguide for a specific word.

The left side is active.

- Select the **Find** tab
- Activate the **Find:** input field
- Enter the desired word and confirm with the ENT key: the TNC lists all sources containing the word
- Use the arrow key to highlight the desired source
- Press the ENT key to go to the selected source

> You can enter the search word only with a keyboard connected via USB.
>
> The full-text search only works for single words.
>
> If you activate the **Search only in titles** function (by mouse or by using the cursor and the space key), the TNC searches only through headings and ignores the body text.

Downloading current help files

You'll find the help files for your TNC software on the HEIDENHAIN homepage **www.heidenhain.de** under:

- Services and Documentation
- Software
- TNC 320 help system
- NC software number of your TNC, for example **34056x-02**
- Select the desired language, for example English: You will see a ZIP file with the appropriate help files
- Download the ZIP file and unzip it
- Move the unzipped CHM files to the TNC in the **TNC:\tncguide\en** directory or into the respective language subdirectory (see also the following table)

If you want to use TNCremoNT to transfer the CHM files to the TNC, then in the **Extras**>**Configuration**>**Mode**>**Transfer in binary format** menu item you have to enter the extension **.CHM**.

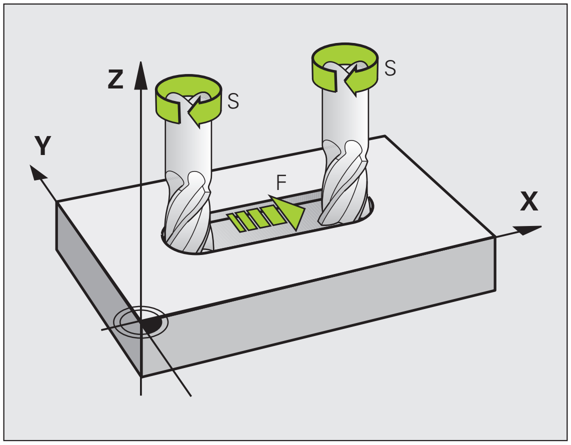
Language	TNC directory
German	**TNC:\tncguide\de**
English	**TNC:\tncguide\en**
Czech	**TNC:\tncguide\cs**
French	**TNC:\tncguide\fr**
Italian	**TNC:\tncguide\it**
Spanish	**TNC:\tncguide\es**
Portuguese	**TNC:\tncguide\pt**
Swedish	**TNC:\tncguide\sv**
Danish	**TNC:\tncguide\da**
Finnish	**TNC:\tncguide\fi**
Dutch	**TNC:\tncguide\nl**
Polish	**TNC:\tncguide\pl**
Hungarian	**TNC:\tncguide\hu**
Russian	**TNC:\tncguide\ru**
Chinese (simplified)	**TNC:\tncguide\zh**
Chinese (traditional)	**TNC:\tncguide\zh-tw**

5

Programming: Tools

5.1 Entering Tool-Related Data

Feed rate F



The feed rate **F** is the speed (in millimeters per minute or inches per minute) at which the tool center point moves. The maximum feed rates can be different for the individual axes and are set in machine parameters.

Input

You can enter the feed rate in the **T** block and in every positioning block (see "Programming tool movements in DIN/ISO format" on page 82). In millimeter-programs you enter the feed rate in mm/min, and in inch-programs, for reasons of resolution, in 1/10 inch/min.

Rapid traverse

If you wish to program rapid traverse, enter **G00**.

Duration of effect

A feed rate entered as a numerical value remains in effect until a block with a different feed rate is reached. If the new feed rate is **G00** (rapid traverse), the last programmed feed rate is once again valid after the next block with **G01**.

Changing during program run

You can adjust the feed rate during program run with the feed-rate override knob F.

Spindle speed S

The spindle speed S is entered in revolutions per minute (rpm) in a **T** block. Instead, you can also define the cutting speed Vc in m/min.

Programmed change

In the part program, you can change the spindle speed in a **T** block by entering the spindle speed only:

SPEC FCT

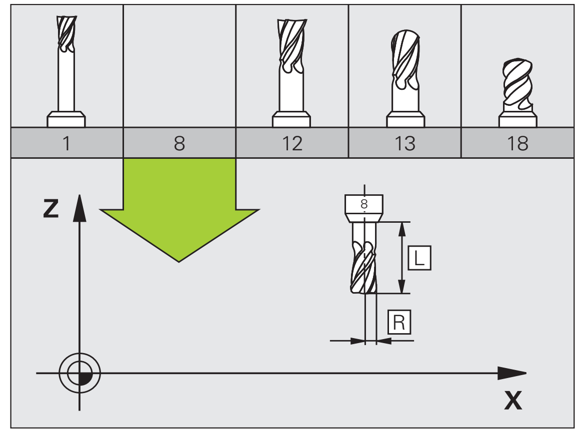
- To program the spindle speed, press the SPEC FCT key.
- Press the PROGRAM FUNCTIONS soft key
- Press the DIN/ISO soft key
- Press the S soft key.
- Enter the new spindle speed

Changing during program run

You can adjust the spindle speed during program run with the spindle speed override knob S.

5.2 Tool Data

Requirements for tool compensation



You usually program the coordinates of path contours as they are dimensioned in the workpiece drawing. To allow the TNC to calculate the tool center path—i.e. the tool compensation—you must also enter the length and radius of each tool you are using.

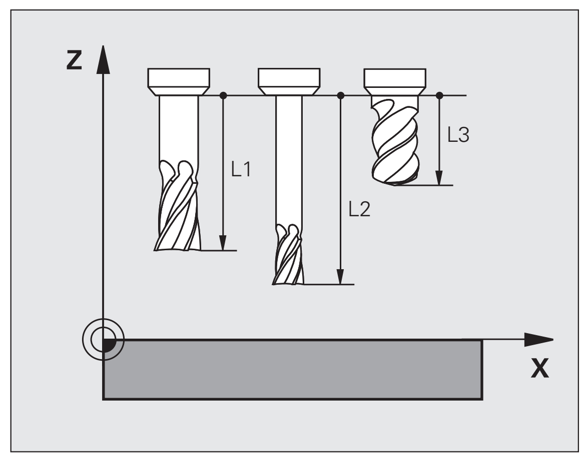
Tool data can be entered either directly in the part program with **G99** or separately in a tool table. In a tool table, you can also enter additional data for the specific tool. The TNC will consider all the data entered for the tool when executing the part program.

Tool numbers and tool names

Each tool is identified by a number between 0 and 32767. If you are working with tool tables, you can also enter a tool name for each tool. Tool names can have up to 16 characters.

The tool number 0 is automatically defined as the zero tool with the length L=0 and the radius R=0. In tool tables, tool T0 should also be defined with L=0 and R=0.

Tool length L

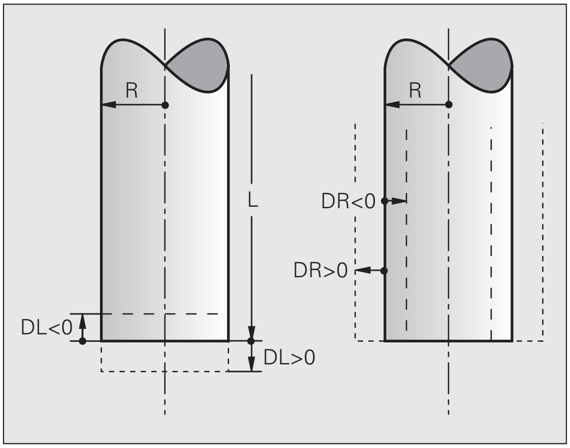


You should always enter the tool length L as an absolute value based on the tool reference point. The entire tool length is essential for the TNC in order to perform numerous functions involving multi-axis machining.

Tool radius R

You can enter the tool radius R directly.

Delta values for lengths and radii



Delta values are offsets in the length and radius of a tool.

A positive delta value describes a tool oversize (**DL, DR, DR2**>0). If you are programming the machining data with an allowance, enter the oversize value in the **T** block of the part program.

A negative delta value describes a tool undersize (**DL, DR, DR2**<0). An undersize is entered in the tool table for wear.

Delta values are usually entered as numerical values. In a **T** block, you can also assign the values to Q parameters.

Input range: You can enter a delta value with up to ± 99.999 mm.

Delta values from the tool table influence the graphical representation of the **tool.** The representation of the **workpiece** remains the same in the simulation.

Delta values from the **T** block change the represented size of the **workpiece** during the simulation. The simulated **tool size** remains the same.

Entering tool data into the program

The number, length and radius of a specific tool is defined in the **G99** block of the part program.

- To select tool definition, press the TOOL DEF key.

TOOL DEF

- **Tool number:** Each tool is uniquely identified by its tool number.
- **Tool length:** Compensation value for the tool length
- **Tool radius:** Compensation value for the tool radius

In the programming dialog, you can transfer the value for tool length and tool radius directly into the input line by pressing the desired axis soft key.

Example

```
N40 G99 T5 L+10 R+5 *
```

Entering tool data in the table

You can define and store up to 9999 tools and their tool data in a tool table. Also see the Editing Functions later in this Chapter. In order to be able to assign various compensation data to a tool (indexing tool number), insert a line and extend the tool number by a dot and a number from 1 to 9 (e.g. **T 5.2**).

You must use tool tables if

- you wish to use indexed tools such as stepped drills with more than one length compensation value,
- your machine tool has an automatic tool changer,
- you want to rough-mill the contour with Cycle G122, (see “User's Manual for Cycle Programming, ROUGH-OUT”).
- you want to work with Cycles 251 to 254 (see "User's Manual for Cycle Programming," Cycles 251 to 254)

> If you create or manage further tool tables, the file name has to start with a letter.

Tool table: Standard tool data

Abbr.	Inputs	Dialog
T	Number by which the tool is called in the program (e.g. 5, indexed: 5.2)	-
NAME	Name by which the tool is called in the program (no more than 16 characters, all capitals, no spaces)	**Tool name?**
L	Compensation value for tool length L	**Tool length?**
R	Compensation value for the tool radius R	**Tool radius R?**
R2	Tool radius R2 for toroid cutters (only for 3-D radius compensation or graphical representation of a machining operation with spherical or toroid cutters)	**Tool radius R2?**
DL	Delta value for tool length L	**Tool length oversize?**
DR	Delta value for tool radius R	**Tool radius oversize?**
DR2	Delta value for tool radius R2	**Tool radius oversize R2?**
LCUTS	Tooth length of the tool for Cycle 22	**Tooth length in the tool axis?**
ANGLE	Maximum plunge angle of the tool for reciprocating plunge-cut in Cycles 22 and 208	**Maximum plunge angle?**
TL	Set tool lock (**TL:** for Tool Locked)	**Tool locked?** **Yes = ENT / No = NO ENT**
RT	Number of a replacement tool, if available (**RT**: for Replacement Tool; see also **TIME2**)	**Replacement tool?**

Abbr.	Inputs	Dialog
TIME1	Maximum tool life in minutes. This function can vary depending on the individual machine tool. Your machine manual provides more information	**Maximum tool age?**
TIME2	Maximum tool life in minutes during **TOOL CALL:** If the current tool age reaches or exceeds this value, the TNC changes the tool during the next **TOOL CALL** (see also **CUR_TIME**).	**Maximum tool age for TOOL CALL?**
CUR_TIME	Current age of the tool in minutes: The TNC automatically counts the current tool life **(CUR_TIME)**. A starting value can be entered for used tools	**Current tool life?**
TYPE	Tool type: Press the SELECT TYPE (3rd soft-key row); the TNC superimposes a window where you can select the type of tool you want. You can assign tool types to specify the display filter settings such that only the selected type is visible in the table.	**Tool type?**
DOC	Comment on tool (up to 16 characters)	**Tool description?**
PLC	Information on this tool that is to be sent to the PLC	**PLC status?**
PTYP	Tool type for evaluation in the pocket table	**Tool type for pocket table?**
LIFTOFF	Definition of whether the TNC should retract the tool in the direction of the positive tool axis at an NC stop in order to avoid leaving dwell marks on the contour. If **Y** is defined, the TNC retracts the tool from the contour, provided that this function was activated in the NC program with M148 (see "Automatically retract tool from the contour at an NC stop: M148" on page 285).	**Retract tool Y/N ?**
TP_NO	Reference to the number of the touch probe in the touch-probe table	**Number of the touch probe**
T_ANGLE	Point angle of the tool. Is used by the Centering cycle (Cycle 240) in order to calculate the centering depth from the diameter entry	**Point angle?**
LAST_USE	Date and time at which the TNC inserted the tool for the last time via **TOOL CALL**. **Input range**: 16 characters max., format internally specified: Date = yyyy.mm.dd, time = hh.mm	**LAST_USE**

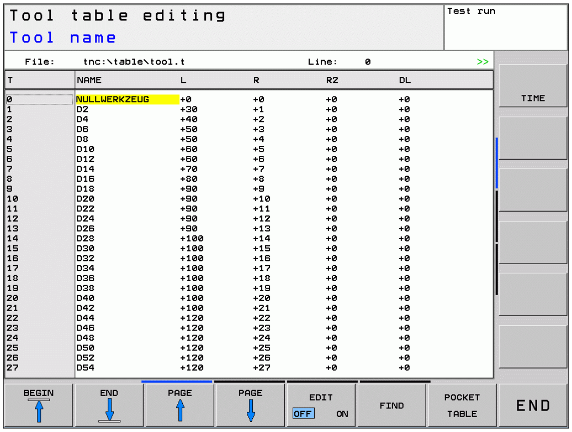
Tool table: Tool data required for automatic tool measurement

For a description of the cycles for automatic tool measurement, see the User's Manual for Cycle Programming.

Abbr.	Inputs	Dialog
`CUT`	Number of teeth (20 teeth maximum)	**`Number of teeth?`**
`LTOL`	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the TNC locks the tool (status **L**). Input range: 0 to 0.9999 mm	**`Wear tolerance: length?`**
`RTOL`	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the TNC locks the tool (status **L**). Input range: 0 to 0.9999 mm	**`Wear tolerance: radius?`**
`R2TOL`	Permissible deviation from tool radius R2 for wear detection. If the entered value is exceeded, the TNC locks the tool (status **L**). Input range: 0 to 0.9999 mm	**`Wear tolerance: Radius 2?`**
`DIRECT.`	Cutting direction of the tool for measuring the tool during rotation	**`Cutting direction (M3 = –)?`**
`R_OFFS`	Tool length measurement: Tool offset between stylus center and tool center. Default setting: No value entered (offset = tool radius)	**`Tool offset: radius?`**
`L_OFFS`	Tool radius measurement: tool offset in addition to **offsetToolAxis** (114104) between upper surface of stylus and lower surface of tool. Default: 0	**`Tool offset: length?`**
`LBREAK`	Permissible deviation from tool length **L** for breakage detection. If the entered value is exceeded, the TNC locks the tool (status **L**). Input range: 0 to 0.9999 mm	**`Breakage tolerance: length?`**
`RBREAK`	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the TNC locks the tool (status **L**). Input range: 0 to 0.9999 mm	**`Breakage tolerance: radius?`**

Editing tool tables

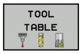
The tool table that is active during execution of the part program is designated TOOL.T and must be saved in the directory **`TNC:\table`**. TOOL.T can only be edited in one of the machine operating modes.



Other tool tables that are to be archived or used for test runs are given any other names with the extension T. By default, for Test Run and Programming modes the TNC uses the "simtool.t" table, which is also stored in the "table" directory. In the Test Run mode, press the TOOL TABLE soft key to edit it.

To open the tool table TOOL.T:

- Select any machine operating mode

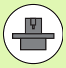


- Press the TOOL TABLE soft key to select the tool table

- Set the EDIT soft key to ON

Display only specific tool types (filter setting)

- Press the TABLE FILTER soft key (fourth soft-key row).
- Select the tool type by pressing a soft key: The TNC only shows tools of the type selected
- Cancel filter: Press the tool type selected before again or select another tool type



The machine tool builder adapts the functional range of the filter function to the requirements of your machine. The machine tool manual provides further information.

To open any other tool table

- Select the Programming and Editing mode of operation

PGM MGT

- Call the file manager
- Press the SELECT TYPE soft key to select the file type
- To show type .T files, press the SHOW .T soft key
- Select a file or enter a new file name. Conclude your entry with the ENT key or the SELECT soft key

When you have opened the tool table, you can edit the tool data by moving the cursor to the desired position in the table with the arrow keys or the soft keys. You can overwrite the stored values, or enter new values at any position. The available editing functions are illustrated in the table below.

If the TNC cannot show all positions in the tool table in one screen page, the highlight bar at the top of the table will display the **>>** or **<<** symbols.

Editing functions for tool tables	Soft key
Select beginning of table	BEGIN
Select end of table	END
Select previous page in table	PAGE
Select next page in table	PAGE
Find the text or number	FIND
Move to beginning of line	BEGIN LINE
Move to end of line	END LINE
Copy highlighted field	COPY FIELD
Insert copied field	PASTE FIELD
Add the entered number of lines (tools) at the end of the table	APPEND N LINES
Insert a line with definable tool number	INSERT LINE
Delete current line (tool)	DELETE LINE

Editing functions for tool tables	Soft key
Sort the tools according to the content of a column	SORT
Show all drills in the tool table	DRILL
Show all cutters in the tool table	CUTTER
Show all taps/thread cutters in the tool table	TAP/ THREAD CUTTER
Show all touch probes in the tool table	TOUCH PROBE

Leaving the tool table

- Call the file manager and select a file of a different type, such as a part program

Importing tool tables

The machine manufacturer can adapt the IMPORT TABLE function. The machine tool manual provides further information.

If you read a tool table from an iTNC 530 and import it into an TNC 320, you have to adapt its format and content before you can use the tool table. On the TNC 320 you can adapt the tool table conveniently with the IMPORT TABLE function. The TNC converts the contents of the imported tool table to a valid format and saves a copy of it under the name TOOL.T. Follow this procedure:

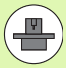
- Save the tool table from the iTNC 530 to the **TNC:\table** directory.
- Select the Programming mode of operation
- To call the file manager, press the PGM MGT key
- Move the highlight to the tool table you want to import
- Press the ADDITIONAL FUNCTIONS soft key
- Press the IMPORT TABLE soft key
- Open the TOOL.T table and check its contents

The following characters are permitted in the **Name** column of the tool table: "ABCDEFGHIJKLMNOPQRSTUVWXYZ0123456789#$&-._". The TNC changes a comma in the tool name to a period during import.

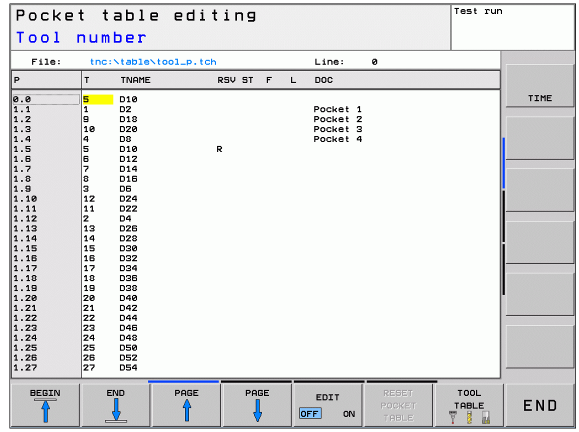
When executing the IMPORT TABLE function, the TNC generates a table with the name TOOL.T. If a file with the same name already exists, it is overwritten. The TNC also creates a backup file with the name TOOL.t.bak. To avoid losing data, be sure to make a backup copy of your original tool table before importing it!

The procedure for copying tool tables using the TNC file manager is described in the section on file management.

Pocket table for tool changer

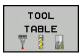


The machine tool builder adapts the functional range of the pocket table to the requirements of your machine. The machine tool manual provides further information.

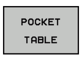


For automatic tool changing you need the pocket table TOOL_P.TCH. The TNC can manage several pocket tables with any file names. To activate a specific pocket table for program run you must select it in the file management of a Program Run mode of operation (status M).

Editing a pocket table in a Program Run operating mode



- Press the TOOL TABLE soft key to select the tool table



- Press the POCKET TABLE soft key to select the pocket table

- Set the EDIT soft key to ON. On your machine this might not be necessary or even possible. Refer to your machine manual

Selecting a pocket table in the Programming and Editing mode of operation

PGM MGT

- Call the file manager
- Press the SHOW ALL soft key to select the file type.
- Select a file or enter a new file name. Conclude your entry with the ENT key or the SELECT soft key

Abbr.	Inputs	Dialog
P	Pocket number of the tool in the tool magazine	-
T	Tool number	**Tool number?**
RSV	Pocket reservation for box magazines	**Pocket reserv.: Yes = ENT / No = NOENT**
ST	Special Tool with a large radius requiring several pockets in the tool magazine. If your special tool takes up pockets in front of and behind its actual pocket, these additional pockets need to be locked in column L (status L)	**Special tool?**
F	Fixed tool number. The tool is always returned to the same pocket in the tool magazine	**Fixed pocket? Yes = ENT / No = NO ENT**
L	Locked pocket (see also column ST)	**Pocket locked Yes = ENT / No = NO ENT**
DOC	Display of the comment to the tool from TOOL.T	-
PLC	Information on this tool pocket that is to be sent to the PLC	**PLC status?**
P1 ... P5	Function is defined by the machine tool builder. The machine tool documentation provides further information.	**Value?**
PTYP	Tool type. Function is defined by the machine tool builder. The machine tool documentation provides further information.	**Tool type for pocket table?**
LOCKED_ABOVE	Box magazine: Lock the pocket above	**Lock the pocket above?**
LOCKED_BELOW	Box magazine: Lock the pocket below	**Lock the pocket below?**
LOCKED_LEFT	Box magazine: Lock the pocket at left	**Lock the pocket at left?**
LOCKED_RIGHT	Box magazine: Lock the pocket at right	**Lock the pocket at right?**

Editing functions for pocket tables	Soft key
Select beginning of table	BEGIN
Select end of table	END
Select previous page in table	PAGE
Select next page in table	PAGE
Reset pocket table	RESET POCKET TABLE
Reset tool number column T	RESET COLUMN T
Go to beginning of the line	BEGIN LINE
Go to end of the line	END LINE
Simulate a tool change	SIMULATED TOOL CHANGE
Select a tool from the tool table: The TNC shows the contents of the tool table. Use the arrow keys to select a tool, press OK to transfer it to the pocket table	SELECT
Edit the current field	EDIT CURRENT FIELD
Sort the view	SORT

The machine manufacturer defines the features, properties and designations of the various display filters. The machine tool manual provides further information.

Calling tool data

A TOOL CALL block in the part program is defined with the following data:

- Select the tool call function with the TOOL CALL key

TOOL CALL

- **`Tool number:`** Enter the number or name of the tool. The tool must already be defined in a **`G99`** block or in the tool table. Press the TOOL NAME soft key to enter the name. The TNC automatically places the tool name in quotation marks. The tool name always refers to the entry in the active tool table TOOL.T. If you wish to call a tool with other compensation values, also enter the index you defined in the tool table after the decimal point. There is a SELECT soft key for calling a window from which you can select a tool defined in the tool table TOOL.T directly without having to enter the number or name.
- **`Working spindle axis X/Y/Z:`** Enter the tool axis
- **`Spindle speed S:`** Enter the spindle speed in rpm Alternatively, you can define the cutting speed Vc in m/min. Press the VC soft key
- **`Feed rate F`**: F [mm/min or 0.1 inch/min] is effective until you program a new feed rate in a positioning or **`T`** block.
- **`Tool length oversize DL:`** Enter the delta value for the tool length
- **`Tool radius oversize DR:`** Enter the delta value for the tool radius
- **`Tool radius oversize DR2:`** Enter the delta value for the tool radius 2

Example: Tool call

Call tool number 5 in the tool axis Z with a spindle speed of 2500 rpm and a feed rate of 350 mm/min. The tool length is to be programmed with an oversize of 0.2 mm, the tool radius 2 with an oversize of 0.05 mm, and the tool radius with an undersize of 1 mm.

```
N20 T 5.2 G17 S2500 DL+0.2 DR-1
```

The character **D** preceding **L** and **R** designates a delta value.

Tool preselection with tool tables

If you are working with tool tables, use **G51** to preselect the next tool. Simply enter the tool number or a corresponding Q parameter, or type the tool name in quotation marks.

Tool change

The tool change function can vary depending on the individual machine tool. The machine tool manual provides further information.

Tool change position

The tool change position must be approachable without collision. With the miscellaneous functions **M91** and **M92,** you can enter machine-based (rather than workpiece-based) coordinates for the tool change position. If **T 0** is programmed before the first tool call, the TNC moves the tool spindle in the tool axis to a position that is independent of the tool length.

Manual tool change

To change the tool manually, stop the spindle and move the tool to the tool change position:

- Move to the tool change position under program control.
- Interrupt program run (see "Interrupting machining", page 384).
- Change the tool.
- Resume program run (see "Resuming program run after an interruption", page 386).

Automatic tool change

If your machine tool has automatic tool changing capability, the program run is not interrupted. When the TNC reaches a **T** it replaces the inserted tool by another from the tool magazine.

Automatic tool change if the tool life expires: M101

The function of **M101** can vary depending on the individual machine tool. The machine tool manual provides further information.

When the specified tool life has expired, the TNC can automatically insert a replacement tool and continue machining with it. Activate the miscellaneous function **M101** for this. **M101** is reset with **M102.**

Enter the respective tool life after which machining is to be continued with a replacement tool in the **TIME2** column of the tool table. In the **CUR_TIME** column the TNC enters the current tool life. If the current tool life is higher than the value entered in the **TIME2** column, a replacement tool will be inserted at the next possible point in the program no later than one minute after expiration of the tool life. The change is made only after the NC block has been completed.

The TNC performs the automatic tool change at a suitable point in the program. The automatic tool change is not performed:

- During execution of machining cycles
- While radius compensation is active (**RR**/**RL**)
- Directly after an approach function **APPR**
- Directly before a departure function **DEP**
- Directly before and after **CHF** and **RND**
- During execution of macros
- During execution of a tool change
- Directly after a **TOOL CALL** or **TOOL DEF**
- During execution of SL cycles

Caution: Danger to the workpiece and tool!

Switch off the automatic tool change with **M102** if you are working with special tools (e.g. side mill cutter) because the TNC at first always moves the tool away from the workpiece in tool axis direction.

Depending on the NC program, the machining time can increase as a result of the tool life verification and calculation of the automatic tool change. You can influence this with the optional input element **BT** (block tolerance)

If you enter the **M101** function, the TNC continues the dialog by requesting the **BT**. Here you define the number of NC blocks (1 - 100) by which the automatic tool change may be delayed. The resulting time period by which the tool change is delayed depends on the content of the NC blocks (e.g. feed rate, path). If you do not define **BT**, the TNC uses the value 1 or, if applicable, a default value defined by the machine manufacturer.

The more you increase the value of **BT**, the smaller will be the effect of an extended program duration through **M101**. Please note that this will delay the automatic tool change!

If you want to reset the current life of a tool (e.g. after changing the indexable inserts), enter the value 0 in the **CUR_TIME** column.

The **M101** function is not available for turning tools and in turning mode.

Tool usage test

The tool usage test function must be enabled by your machine manufacturer. Refer to your machine manual.

In order to run a tool usage test, the complete plain-language program must have been simulated in the **Test Run** mode.

Applying the tool usage test

Before starting a program in the Program Run mode of operation, you can use the TOOL USAGE and TOOL USAGE TEST soft keys to check whether the tools being used in the selected program are available and have sufficient remaining service life. The TNC then compares the actual service-life values in the tool table with the nominal values from the tool usage file.

After you have pressed the TOOL USAGE TEST soft key, the TNC displays the result of the tool usage test in a pop-up window. To close the pop-up window, press the ENT key.

The TNC saves the tool usage times in a separate file with the extension **pgmname.H.T.DEP**. The generated tool usage file contains the following information:

Column	Meaning
TOKEN	■ **TOOL:** Tool usage time per **TOOL CALL.** The entries are listed in chronological order. ■ **TTOTAL**: Total usage time of a tool ■ **STOTAL:** Call of a subprogram; the entries are listed in chronological order ■ **TIMETOTAL:** The total machining time of the NC program is entered in the **WTIME** column. In the **PATH** column the TNC saves the path name of the corresponding NC programs. The **TIME** column shows the sum of all **TIME** entries (without rapid traverse). The TNC sets all other columns to 0 ■ **TOOLFILE:** In the **PATH** column, the TNC saves the path name of the tool table with which you conducted the Test Run. This enables the TNC during the actual tool usage test to detect whether you performed the test run with the TOOL.T.
TNR	Tool number (**–1**: No tool inserted yet)
IDX	Tool index
NAME	Tool name from the tool table
TIME	Tool-usage time in seconds (feed time)
WTIME	Tool-usage time in seconds (total usage time between tool changes)

Column	Meaning
RAD	**Tool radius R** + **Oversize of tool radius DR** from the tool table (in mm).
BLOCK	Block number in which the **TOOL CALL** block was programmed
PATH	■ **TOKEN** = **TOOL**: Path name of the active main program or subprogram ■ **TOKEN** = **STOTAL:** Path name of the subprogram
T	Tool number with tool index
OVRMAX	Maximum feed rate override that occurred during machining. During test run, the TNC enters the value 100 (%)
OVRMIN	Minimum feed rate override that occurred during machining. During test run, the TNC enters the value -1
NAMEPROG	■ **0**: The tool number is programmed ■ **1**: The tool name is programmed

There are two ways to run a tool usage test for a pallet file:

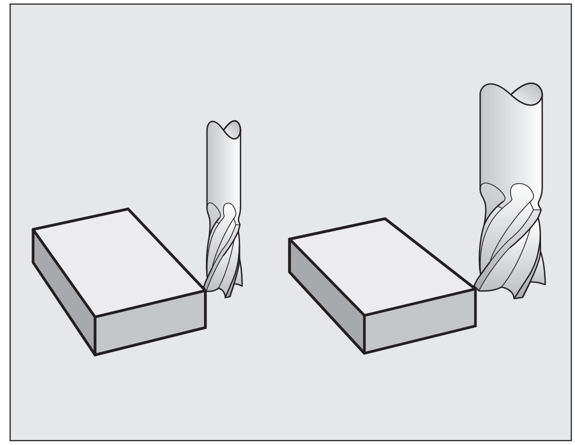
- The highlight is on a pallet entry in the pallet file:
  The TNC runs the tool usage test for the entire pallet.
- The highlight is on a program entry in the pallet file:
  The TNC runs the tool usage test only for the selected program.

5.3 Tool compensation

Introduction

The TNC adjusts the spindle path in the spindle axis by the compensation value for the tool length. In the working plane, it compensates the tool radius.

If you are writing the part program directly on the TNC, the tool radius compensation is effective only in the working plane. The TNC accounts for the compensation value in up to five axes including the rotary axes.



Tool length compensation

Length compensation becomes effective automatically as soon as a tool is called and the spindle axis moves. To cancel length compensation, call a tool with the length L=0.

Caution: Danger of collision!

If you cancel a positive length compensation with **T 0** the distance between tool and workpiece will be reduced.

After **T** the path of the tool in the spindle axis, as entered in the part program, is adjusted by the difference between the length of the previous tool and that of the new one.

For tool length compensation, the control takes the delta values from both the **T** block and the tool table into account:

Compensation value = **L** + $\mathbf{DL}_{\text{TOOL CALL}}$ + $\mathbf{DL}_{\text{TAB}}$ where

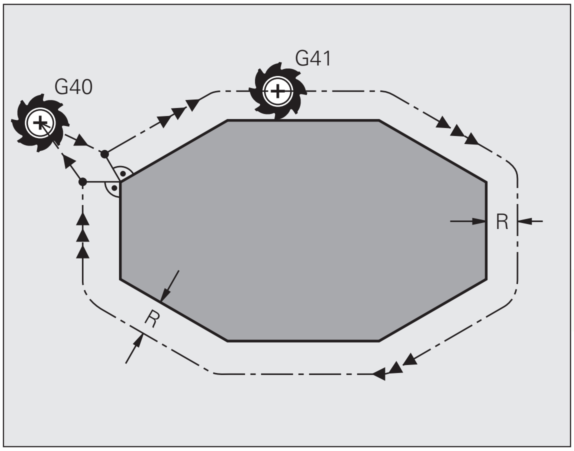
L:	is the tool length **L** from the **G99** block or tool table
$\mathbf{DL}_{\text{TOOL CALL}}$	is the oversize for length **DL** in the **T 0** block (not taken into account by the position display).
$\mathbf{DL}_{\text{TAB}}$	is the oversize for length **DL** in the tool table.

Tool radius compensation

The NC block for programming a tool movement contains:

- **G41** or **G42** for radius compensation
- **G43** or **G44**, for radius compensation in single-axis movements
- **G40** if there is no radius compensation

Radius compensation becomes effective as soon as a tool is called and is moved with a straight line block in the working plane with **G41** or **G42**.



The TNC automatically cancels radius compensation if you:

- program a straight line block with **G40**
- program a **PGM CALL**
- select a new program with PGM MGT.

For radius compensation, the TNC takes the delta values from both the **T** block and the tool table into account:

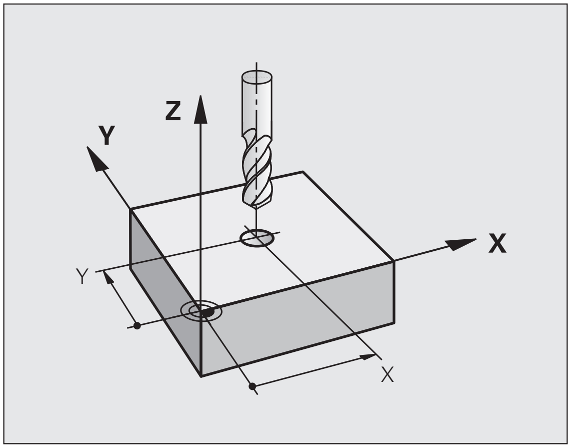
Compensation value = **R** + **DR**$_{\text{TOOL CALL}}$ + **DR**$_{\text{TAB}}$ where

R	Tool radius **R** from the **G99** block or tool table
DR $_{\text{TOOL CALL}}$	Oversize for radius **DR** in the **T** block (not taken into account by the position display)
DR $_{\text{TAB:}}$	Oversize for radius **DR** in the tool table

Contouring without radius compensation: G40

The tool center moves in the working plane along the programmed path or to the programmed coordinates.

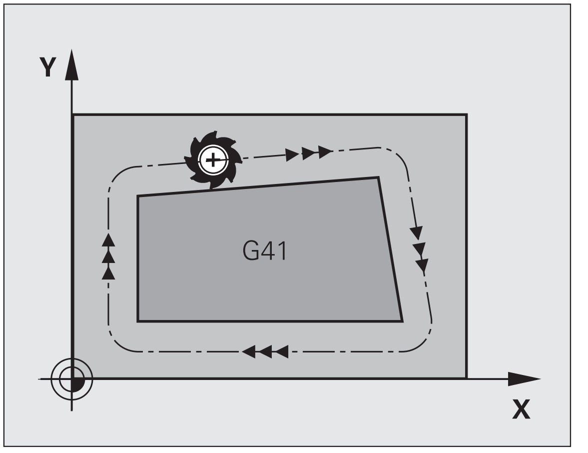
Applications: Drilling and boring, pre-positioning.

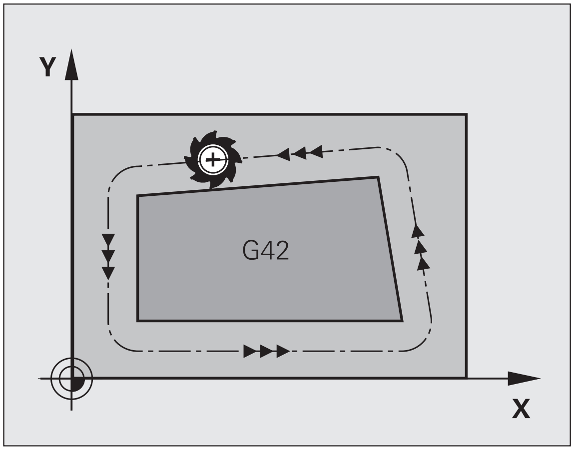


Contouring with radius compensation: G42 and G41

G43 The tool moves to the right of the programmed contour
G42 The tool moves to the left of the programmed contour

The tool center moves along the contour at a distance equal to the radius. "Right" or "left" are to be understood as based on the direction of tool movement along the workpiece contour. See figures.





Between two program blocks with different radius compensations **G43** and **G42** you must program at least one traversing block in the working plane without radius compensation (that is, with **G40**).

The TNC does not put radius compensation into effect until the end of the block in which it is first programmed.

In the first block in which radius compensation is activated with **G42/G41** or canceled with **G40** the TNC always positions the tool perpendicular to the programmed starting or end position. Position the tool at a sufficient distance from the first or last contour point to prevent the possibility of damaging the contour.

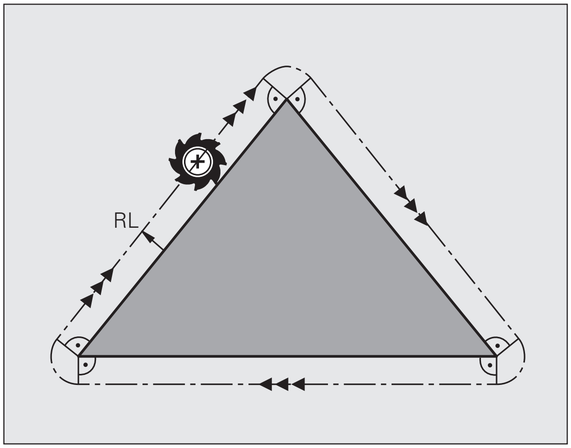
Entering radius compensation

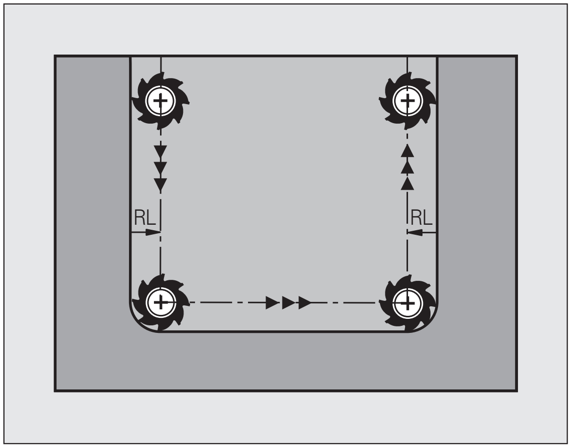
Radius compensation is entered in a G01 block:

G41	To select tool movement to the left of the programmed contour, select function G41, or
G42	To select tool movement to the right of the contour, select function G42, or
G40	To select tool movement without radius compensation or to cancel radius compensation, select function G40
END	To terminate the block, press the END key

Radius compensation: Machining corners

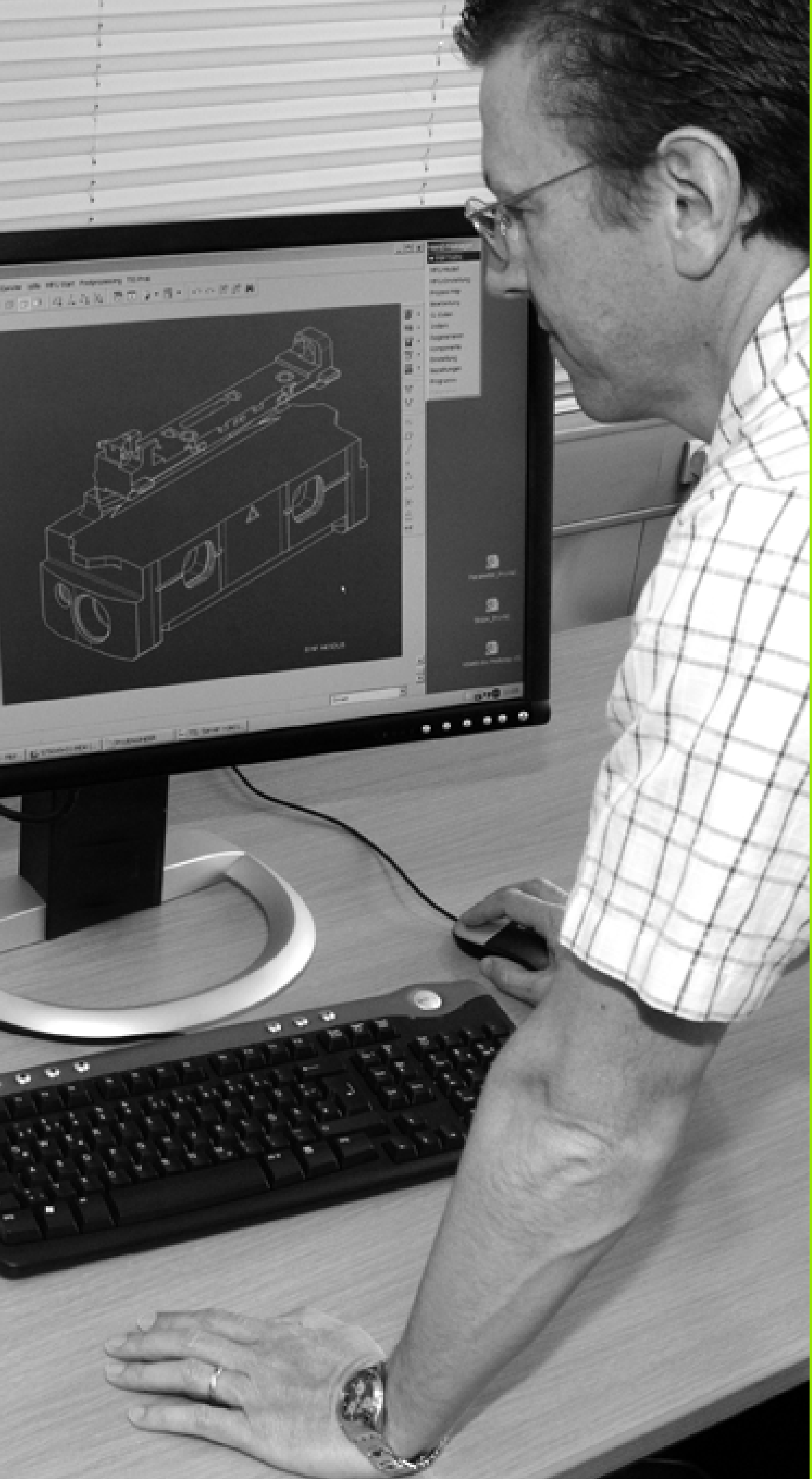
- Outside corners:
  If you program radius compensation, the TNC moves the tool around outside corners on a transitional arc. If necessary, the TNC reduces the feed rate at outside corners to reduce machine stress, for example at very great changes of direction.
- Inside corners:
  The TNC calculates the intersection of the tool center paths at inside corners under radius compensation. From this point it then starts the next contour element. This prevents damage to the workpiece at the inside corners. The permissible tool radius, therefore, is limited by the geometry of the programmed contour.





Danger of collision!

To prevent the tool from damaging the contour, be careful not to program the starting or end position for machining inside corners at a corner of the contour.



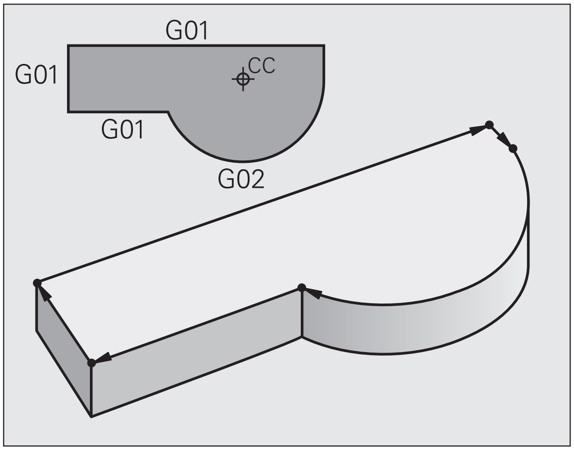
6

Programming: Programming Contours

6.1 Tool Movements

Path functions

A workpiece contour is usually composed of several contour elements such as straight lines and circular arcs. With the path functions, you can program the tool movements for **straight lines** and **circular arcs**.



Miscellaneous functions M

With the TNC's miscellaneous functions you can affect

- The program run, e.g., a program interruption
- The machine functions, such as switching spindle rotation and coolant supply on and off
- The path behavior of the tool

Subprograms and program section repeats

If a machining sequence occurs several times in a program, you can save time and reduce the chance of programming errors by entering the sequence once and then defining it as a subprogram or program section repeat. If you wish to execute a specific program section only under certain conditions, you also define this machining sequence as a subprogram. In addition, you can have a part program call a separate program for execution.

Programming with subprograms and program section repeats is described in Chapter 7.

Programming with Q parameters

Instead of programming numerical values in a part program, you enter markers called Q parameters. You assign the values to the Q parameters separately with the Q parameter functions. You can use the Q parameters for programming mathematical functions that control program execution or describe a contour.

In addition, parametric programming enables you to measure with the 3-D touch probe during program run.

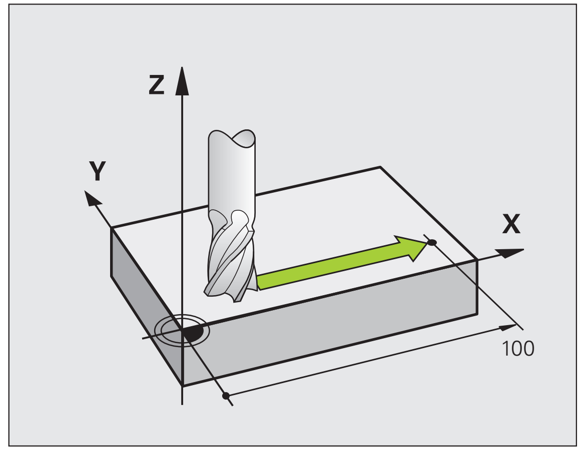
Programming with Q parameters is described in Chapter 8.

6.2 Fundamentals of Path Functions

Programming tool movements for workpiece machining

You create a part program by programming the path functions for the individual contour elements in sequence. You usually do this by entering **the coordinates of the end points of the contour elements** given in the production drawing. The TNC calculates the actual path of the tool from these coordinates, and from the tool data and radius compensation.

The TNC moves all axes programmed in a single block simultaneously.



Movement parallel to the machine axes

The program block contains only one coordinate. The TNC thus moves the tool parallel to the programmed axis.

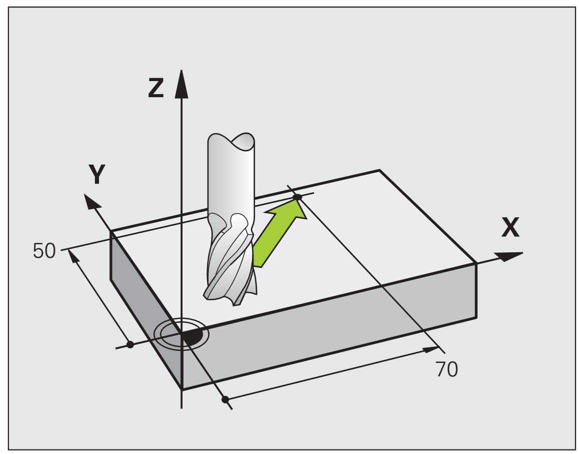
Depending on the individual machine tool, the part program is executed by movement of either the tool or the machine table on which the workpiece is clamped. Nevertheless, you always program path contours as if the tool were moving and the workpiece remaining stationary.

Example:

```
N50 G00 X+100 *
```

N50	Block number
G00	Path function "straight line at rapid traverse"
X+100	Coordinate of the end point

The tool retains the Y and Z coordinates and moves to the position X=100. See figure.



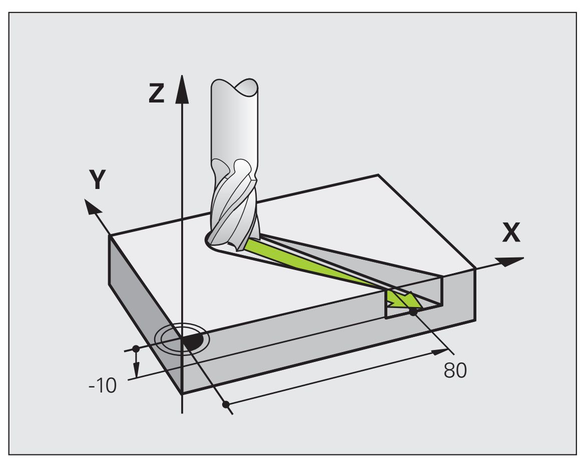
Movement in the main planes

The program block contains two coordinates. The TNC thus moves the tool in the programmed plane.

Example:

```
N50 G00 X+70 Y+50 *
```

The tool retains the Z coordinate and moves in the XY plane to the position X=70, Y=50 (see figure).



Three-dimensional movement

The program block contains three coordinates. The TNC thus moves the tool in space to the programmed position.

Example:

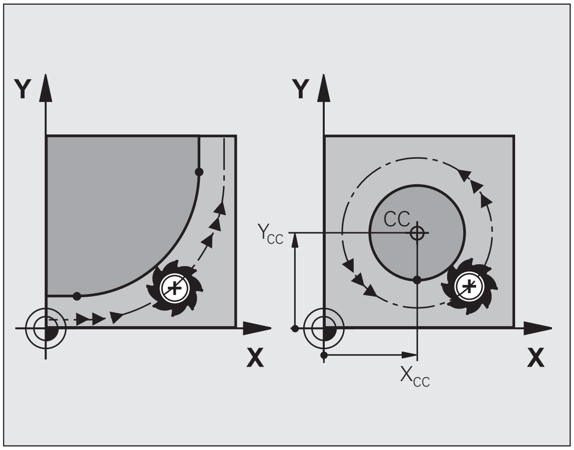
```
N50 G01 X+80 Y+0 Z-10 *
```

Circles and circular arcs

The TNC moves two axes simultaneously on a circular path relative to the workpiece. You can define a circular movement by entering the circle center CC.

When you program a circle, the control assigns it to one of the main planes. This plane is defined automatically when you set the spindle axis during a TOOL CALL:

Spindle axis	Main plane
(G17)	**XY**, also UV, XV, UY
(G18)	**ZX**, also WU, ZU, WX
(G19)	**YZ**, also VW, YW, VZ

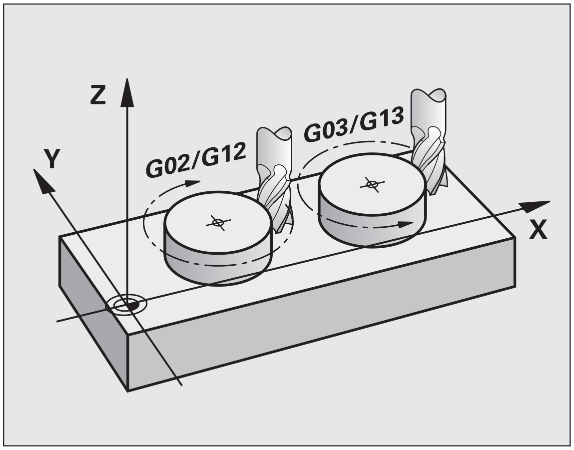


You can program circles that do not lie parallel to a main plane by using the function for tilting the working plane (see User's Manual for Cycles, Cycle 19, WORKING PLANE) or Q parameters (see "Principle and Overview", page 202).

Direction of rotation DR for circular movements

When a circular path has no tangential transition to another contour element, enter the direction of rotation as follows:

Clockwise direction of rotation: **G02/G12**
Counterclockwise direction of rotation: **G03/G13**



Radius compensation

The radius compensation must be in the block in which you move to the first contour element. You cannot activate radius compensation in a circle block. Activate it beforehand in a straight-line block (see "Path Contours—Cartesian Coordinates", page 163).

Pre-positioning

Danger of collision!

Before running a part program, always pre-position the tool to prevent the possibility of damaging it or the workpiece.

6.3 Contour Approach and Departure

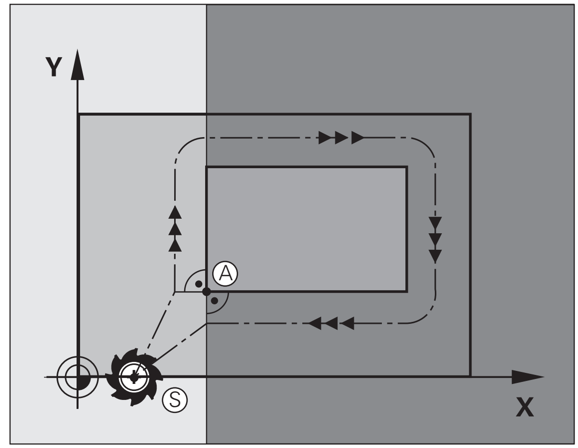
Starting point and end point

The tool approaches the first contour point from the starting point. The starting point must be:

- Programmed without radius compensation
- Approachable without danger of collision
- Close to the first contour point

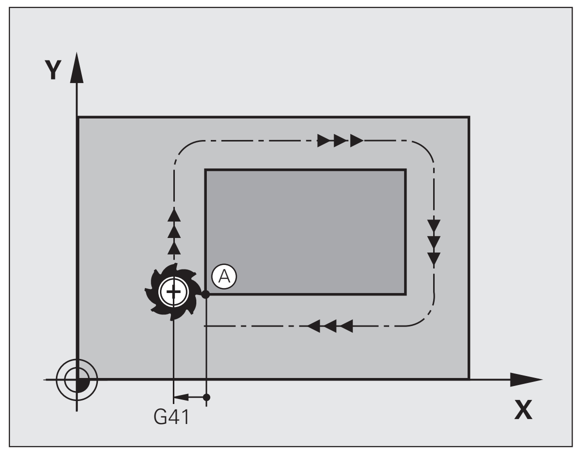
Example

Figure at upper right: If you set the starting point in the dark gray area, the contour will be damaged when the first contour element is approached.



First contour point

You need to program a radius compensation for the tool movement to the first contour point.

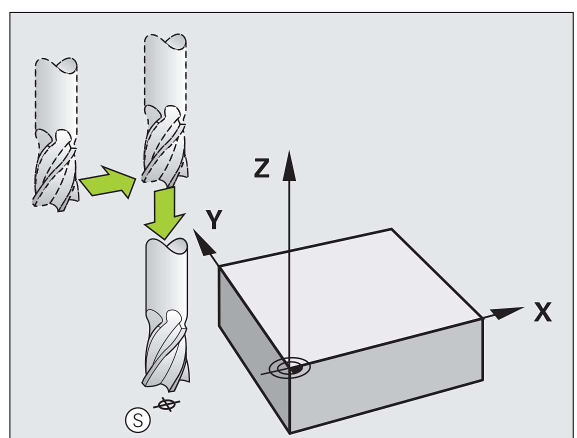


Approaching the starting point in the spindle axis

When the starting point is approached, the tool must be moved to the working depth in the spindle axis. If danger of collision exists, approach the starting point in the spindle axis separately.

Example NC blocks

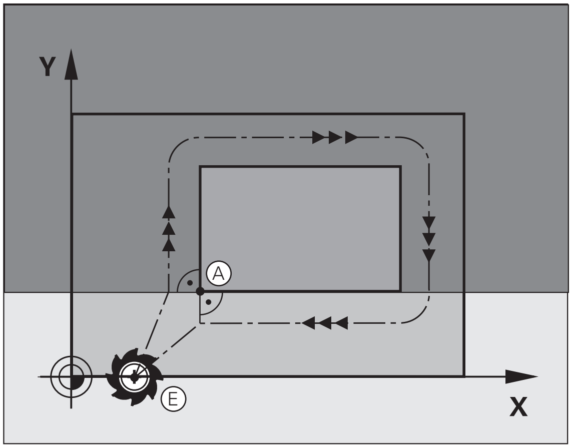
```
N30 G00 G40 X+20 Y+30 *
N40 Z-10 *
```



End point

The end point should be selected so that it is:

- Approachable without danger of collision
- Near to the last contour point
- In order to make sure the contour will not be damaged, the optimal ending point should lie on the extended tool path for machining the last contour element

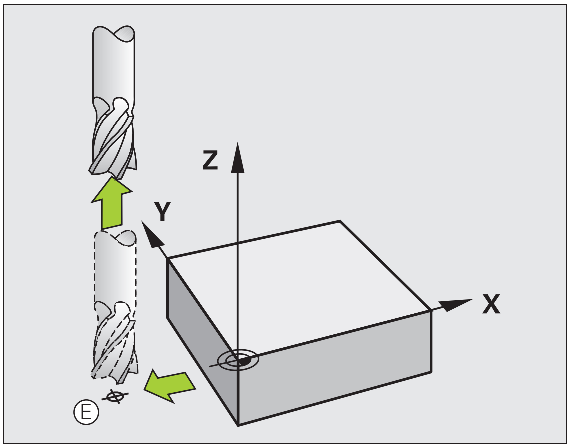


Example

Figure at upper right: If you set the ending point in the dark gray area, the contour will be damaged when the end point is approached.

Depart the end point in the spindle axis:

Program the departure from the end point in the spindle axis separately. See figure at center right.



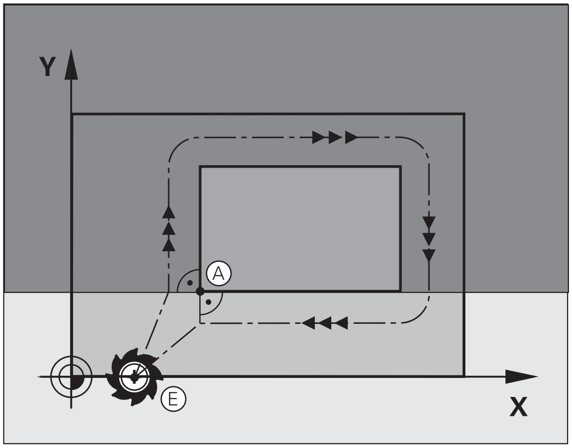
Example NC blocks

```
N50 G00 G40 X+60 Y+70 *
N60 Z+250 *
```

Common starting and end points

Do not program any radius compensation if the starting point and end point are the same.

In order to make sure the contour will not be damaged, the optimal starting point should lie between the extended tool paths for machining the first and last contour elements.

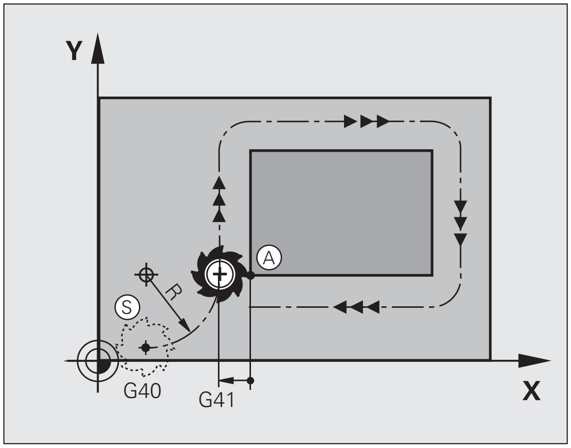


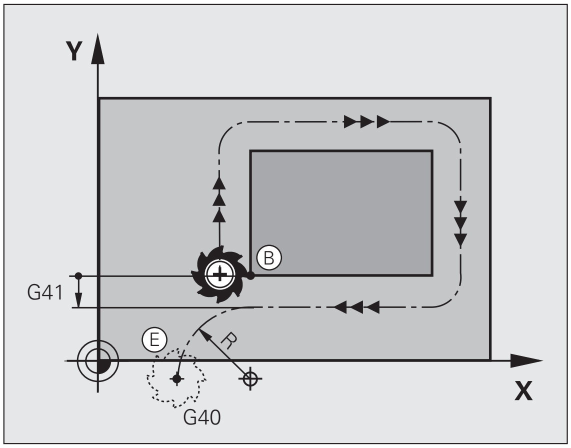
Example

Figure at upper right: If you set the starting point in the dark gray area, the contour will be damaged when the first contour element is approached.

Tangential approach and departure

With **G26** (figure at center right), you can program a tangential approach to the workpiece, and with **G27** (figure at lower right) a tangential departure. In this way you can avoid dwell marks.





Starting point and end point

The starting point and the end point lie outside the workpiece, close to the first and last contour points. They are to be programmed without radius compensation.

Approach

- **G26** is entered after the block in which the first contour element is programmed: This will be the first block with radius compensation **G41/G42**

Departure

- **G27** after the block in which the last contour element is programmed: This will be the last block with radius compensation **G41/G42**

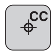
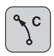
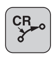
The radius for **G26** and **G27** must be selected so that the TNC can execute the circular path between the starting point and the first contour point, as well as the last contour point and the end point.

Example NC blocks

N50 G00 G40 G90 X-30 Y+50 *	Starting point
N60 G01 G41 X+0 Y+50 F350 *	First contour point
N70 G26 R5 *	Tangential approach with radius R = 5 mm
. . .	
PROGRAM CONTOUR BLOCKS	
. . .	Last contour point
N210 G27 R5 *	Tangential departure with radius R = 5 mm
N220 G00 G40 X-30 Y+50 *	End point

6.4 Path Contours—Cartesian Coordinates

Overview of path functions

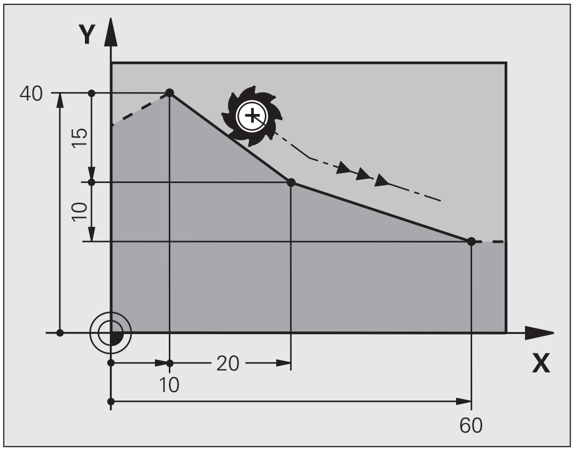
Function	Path function key	Tool movement	Required input	Page
Line **L**		Straight line	Coordinates of the end points of the straight line	Page 164
Chamfer **CHF**		Chamfer between two straight lines	Chamfer side length	Page 165
Circle Center **CC**		None	Coordinates of the circle center or pole	Page 167
Circle **C**		Circular arc around a circle center CC to an arc end point	Coordinates of the arc end point, direction of rotation	Page 168
Circular arc **CR**		Circular arc with a certain radius	Coordinates of the arc end point, arc radius, direction of rotation	Page 169
Circular arc **CT**		Circular arc with tangential connection to the preceding and subsequent contour elements	Coordinates of the arc end point	Page 171
Corner Rounding **RND**		Circular arc with tangential connection to the preceding and subsequent contour elements	Rounding radius R	Page 166

Programming path functions

You can program path functions conveniently by using the gray path function keys. In further dialogs, you are prompted by the TNC to make the required entries.

If you enter DIN/ISO functions via a connected USB keyboard, make sure that capitalization is active.

Straight line at rapid traverse G00
Straight line with feed rate G01 F



The TNC moves the tool in a straight line from its current position to the straight-line end point. The starting point is the end point of the preceding block.

- **Coordinates** of the end point of the straight line, if necessary
- **Radius compensation G40/G41/G42**
- **Feed rate F**
- **Miscellaneous function M**

Movement at rapid traverse

You can also use the L key to create a straight line block for a rapid traverse movement (**G00** block):

- Press the L key to open a program block for a linear movement
- Press the left arrow key to switch to the input range for G codes
- Press the G0 soft key if you want to enter a rapid traverse movement

Example NC blocks

```
N70 G01 G41 X+10 Y+40 F200 M3 *
N80 G91 X+20 Y-15  *
N90 G90 X+60 G91 Y-10 *
```

Actual position capture

You can also generate a straight-line block (**G01** block) by using the ACTUAL-POSITION-CAPTURE key:

- In the Manual Operation mode, move the tool to the position you want to capture
- Switch the screen display to Programming and Editing
- Select the program block after which you want to insert the L block

- Press the ACTUAL-POSITION-CAPTURE key: The TNC generates an L block with the actual position coordinates

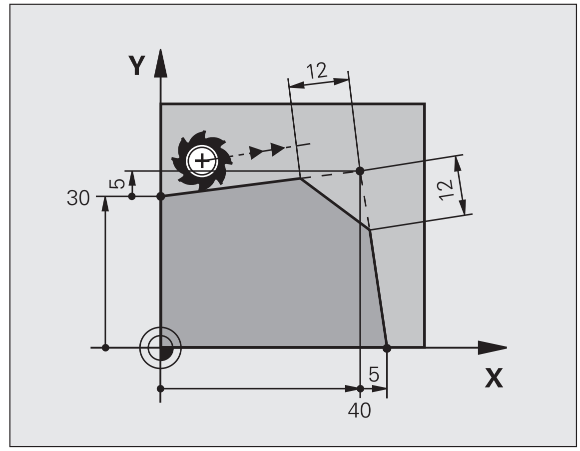
Inserting a chamfer between two straight lines

The chamfer enables you to cut off corners at the intersection of two straight lines.

- The line blocks before and after the **G24** block must be in the same working plane as the chamfer
- The radius compensation before and after the **G24** block must be the same
- The chamfer must be machinable with the current tool

CHF

- **Chamfer side length:** Length of the chamfer, and if necessary:
- **Feed rate F** (effective only in **G24** block)



Example NC blocks

```
N70 G01 G41 X+0 Y+30 F300 M3 *
N80 X+40 G91 Y+5 *
N90 G24 R12 F250 *
N100 G91 X+5 G90 Y+0 *
```

You cannot start a contour with a **G24** block.

A chamfer is possible only in the working plane.

The corner point is cut off by the chamfer and is not part of the contour.

A feed rate programmed in the CHF block is effective only in that block. After the **G24** block, the previous feed rate becomes effective again.

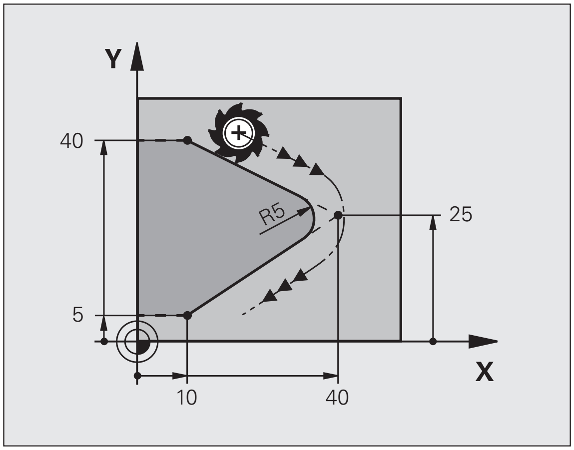
Corner rounding G25

The **G25** function is used for rounding off corners.

The tool moves on an arc that is tangentially connected to both the preceding and subsequent contour elements.

The rounding arc must be machinable with the called tool.

- **Rounding radius:** Enter the radius, and if necessary:
- **Feed rate F** (effective only in **G25** block)



Example NC blocks

```
5 L X+10 Y+40 RL F300 M3
6 L X+40 Y+25
7 RND R5 F100
8 L X+10 Y+5
```

In the preceding and subsequent contour elements, both coordinates must lie in the plane of the rounding arc. If you machine the contour without tool-radius compensation, you must program both coordinates in the working plane.

The corner point is cut off by the rounding arc and is not part of the contour.

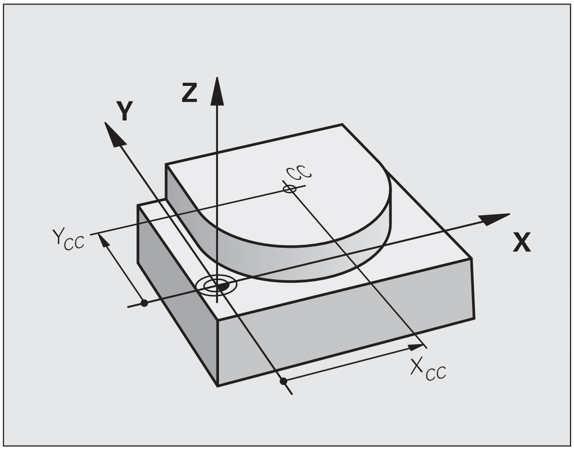
A feed rate programmed in the **G25** block is effective only in that **G25** block. After the **G25** block, the previous feed rate becomes effective again.

You can also use an RND block for a tangential contour approach.

Circle center I, J

You can define a circle center for circles that you have programmed with the **G02**, **G03** or **G05** function. This is done in the following ways:

- Entering the Cartesian coordinates of the circle center in the working plane, or
- Using the circle center defined in an earlier block, or
- Capturing the coordinates with the ACTUAL-POSITION-CAPTURE key



SPEC FCT

- ▶ To program the circle center, press the SPEC FCT key
- ▶ Press the PROGRAM FUNCTIONS soft key
- ▶ Press the DIN/ISO soft key
- ▶ Press the I or J soft key
- ▶ Enter the coordinates for the circle center, or
  If you want to use the last programmed position, enter **G29**

Example NC blocks

```
N50 I+25 J+25 *
```

or

```
N10 G00 G40 X+25 Y+25 *
N20 G29 *
```

The program blocks 10 and 11 do not refer to the illustration.

Duration of effect

The circle center definition remains in effect until a new circle center is programmed. You can also define a circle center for the secondary axes U, V and W.

Entering the circle center incrementally

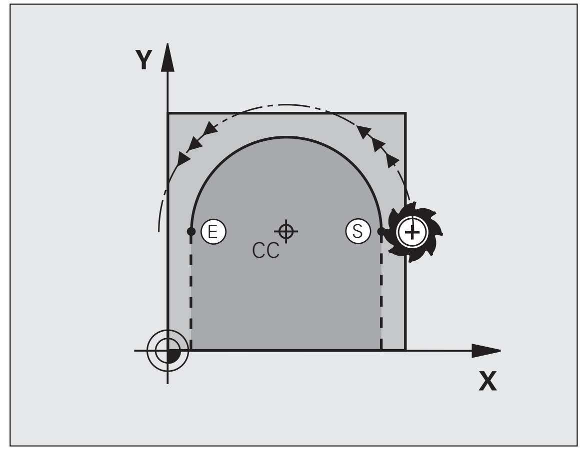
If you enter the circle center with incremental coordinates, you have programmed it relative to the last programmed position of the tool.

The only effect of CC is to define a position as circle center: The tool does not move to this position.

The circle center is also the pole for polar coordinates.

Circular path C around circle center CC

Before programming a circular arc, you must first enter the circle center **I, J**. The last programmed tool position will be the starting point of the arc.



Direction of rotation

- In clockwise direction: **G02**
- In counterclockwise direction: **G03**
- Without programmed direction: **G05.** The TNC traverses the circular arc with the last programmed direction of rotation

▶ Move the tool to the circle starting point

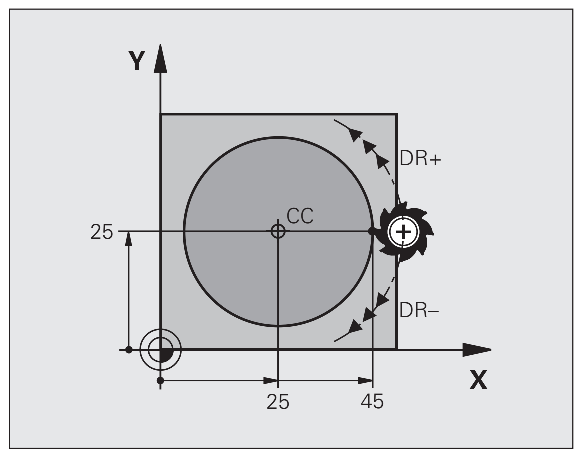
▶ Enter the **coordinates** of the circle center

▶ Enter the **coordinates** of the arc end point, and if necessary:

▶ **Feed rate F**

▶ **Miscellaneous function M**

The TNC normally makes circular movements in the active working plane. If you program circular arcs that do not lie in the active working plane, for example **G2 Z... X...** with a tool axis Z, and at the same time rotate this movement, then the TNC moves the tool in a spatial arc, which means a circular arc in 3 axes.



Example NC blocks

```
N50 I+25 J+25 *
N60 G01 G42 X+45 Y+25 F200 M3 *
N70 G03 X+45 Y+25 *
```

Full circle

For the end point, enter the same point that you used for the starting point.

The starting and end points of the arc must lie on the circle.

Input tolerance: up to 0.016 mm (selected through the **circleDeviation** machine parameter).

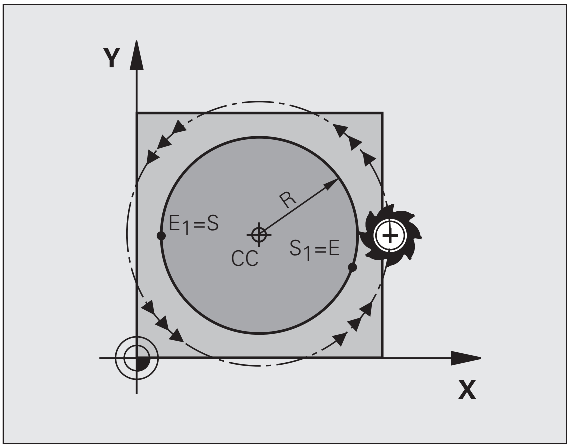
Smallest possible circle that the TNC can traverse: 0.0016 µm.

Circular path G02/G03/G05 with defined radius

The tool moves on a circular path with the radius R.

Direction of rotation

- In clockwise direction: **G02**
- In counterclockwise direction: **G03**
- Without programmed direction: **G05.** The TNC traverses the circular arc with the last programmed direction of rotation



- **Coordinates** of the arc end point
- **Radius R**
  Note: The algebraic sign determines the size of the arc!
- **Miscellaneous function M**
- **Feed rate F**

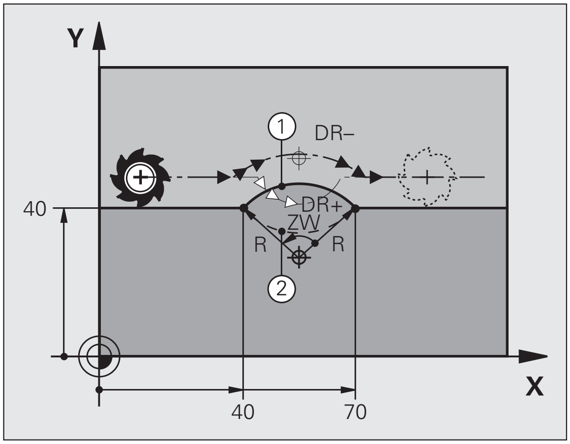
Full circle

For a full circle, program two blocks in succession:

The end point of the first semicircle is the starting point of the second. The end point of the second semicircle is the starting point of the first.

Central angle CCA and arc radius R

The starting and end points on the contour can be connected with four arcs of the same radius:



Smaller arc: CCA<180°
Enter the radius with a positive sign R>0

Larger arc: CCA>180°
Enter the radius with a negative sign R<0

The direction of rotation determines whether the arc is curving outward (convex) or curving inward (concave):

Convex: Direction of rotation **G02** (with radius compensation **G41**)

Concave: Direction of rotation **G03** (with radius compensation **G41**)

Example NC blocks

```
N100 G01 G41 X+40 Y+40 F200 M3 *
N110 G02 X+70 Y+40 R+20 * (ARC 1)
```

or

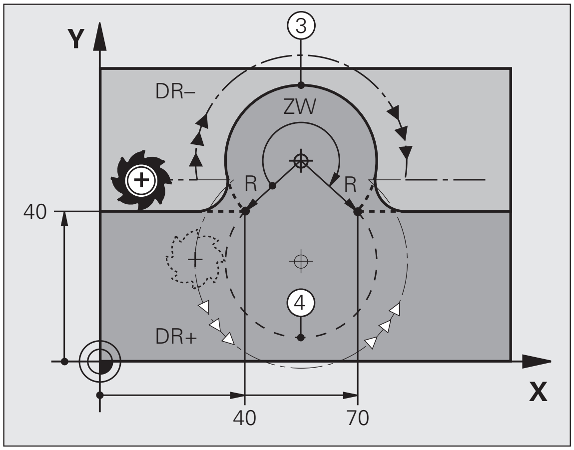
```
N110 G03 X+70 Y+40 R+20 * (ARC 2)
```

or

```
N110 G02 X+70 Y+40 R-20 * (ARC 3)
```

or

```
N110 G03 X+70 Y+40 R-20 * (ARC 4)
```



The distance from the starting and end points of the arc diameter cannot be greater than the diameter of the arc.

The maximum radius is 99.9999 m.

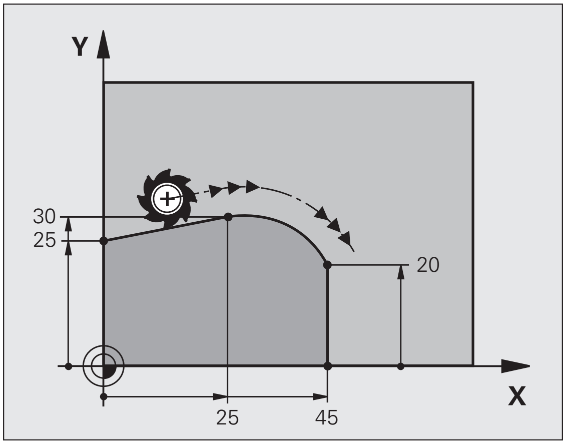
You can also enter rotary axes A, B and C.

Circular path G06 with tangential connection

The tool moves on an arc that starts tangentially to the previously programmed contour element.

A transition between two contour elements is called tangential when there is no kink or corner at the intersection between the two contours—the transition is smooth.

The contour element to which the tangential arc connects must be programmed immediately before the **G06** block. This requires at least two positioning blocks.



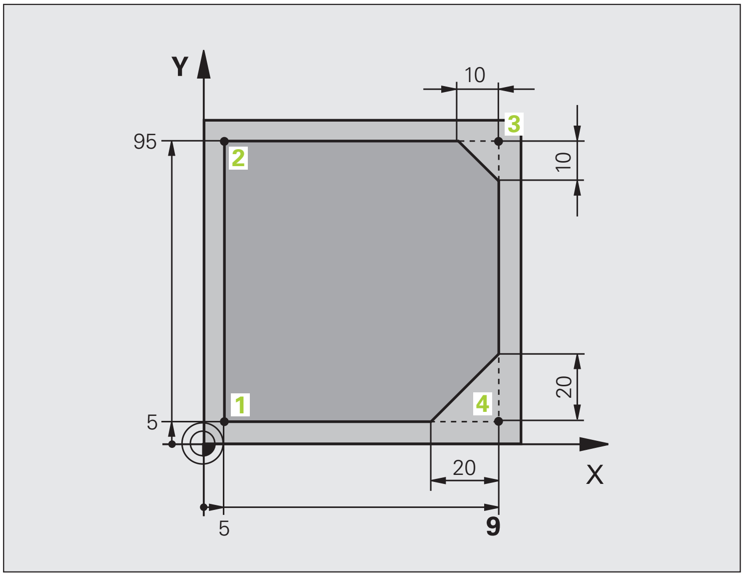
- **Coordinates** of the arc end point, and if necessary:
- **Feed rate F**
- **Miscellaneous function M**

Example NC blocks

```
N70 G01 G41 X+0 Y+25 F300 M3 *
N80 X+25 Y+30 *
N90 G06 X+45 Y+20 *
G01 Y+0 *
```

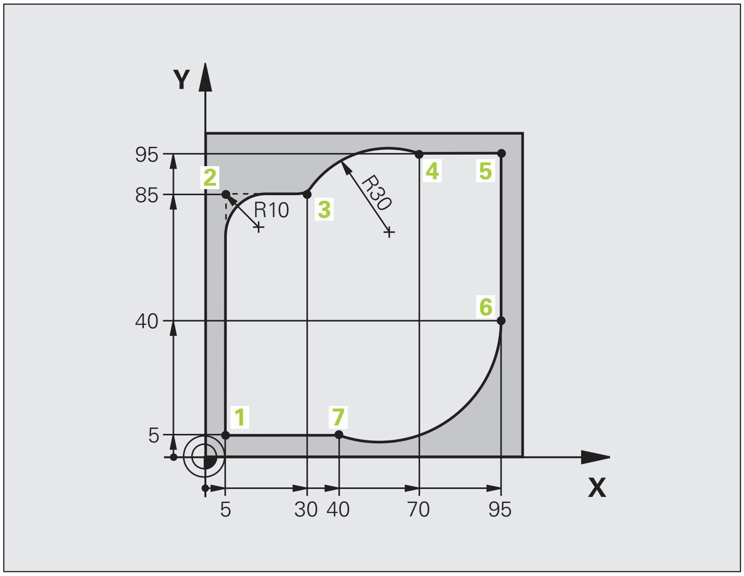
A tangential arc is a two-dimensional operation: the coordinates in the **G06** block and in the contour element preceding it must be in the same plane of the arc!

Example: Linear movements and chamfers with Cartesian coordinates



%LINEAR G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define blank form for graphic workpiece simulation
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 T1 G17 S4000 *	Call tool in the spindle axis and with the spindle speed S
N40 G00 G40 G90 Z+250 *	Retract tool in the spindle axis at rapid traverse
N50 X-10 Y-10 *	Pre-position the tool
N60 G01 Z-5 F1000 M3 *	Move to working depth at feed rate F = 1000 mm/min
N70 G01 G41 X+5 Y+5 F300 *	Approach the contour at point 1, activate radius compensation G41
N80 G26 R5 F150 *	Tangential approach
N90 Y+95 *	Move to point 2
N100 X+95 *	Point 3: first straight line for corner 3
N110 G24 R10 *	Program chamfer with length 10 mm
N120 Y+5 *	Point 4: 2nd straight line for corner 3, 1st straight line for corner 4
N130 G24 R20 *	Program chamfer with length 20 mm
N140 X+5 *	Move to last contour point 1, second straight line for corner 4
N150 G27 R5 F500 *	Tangential exit
N160 G40 X-20 Y-20 F1000 *	Retract tool in the working plane, cancel radius compensation
N170 G00 Z+250 M2 *	Retract in the tool axis, end program
N99999999 %LINEAR G71 *	

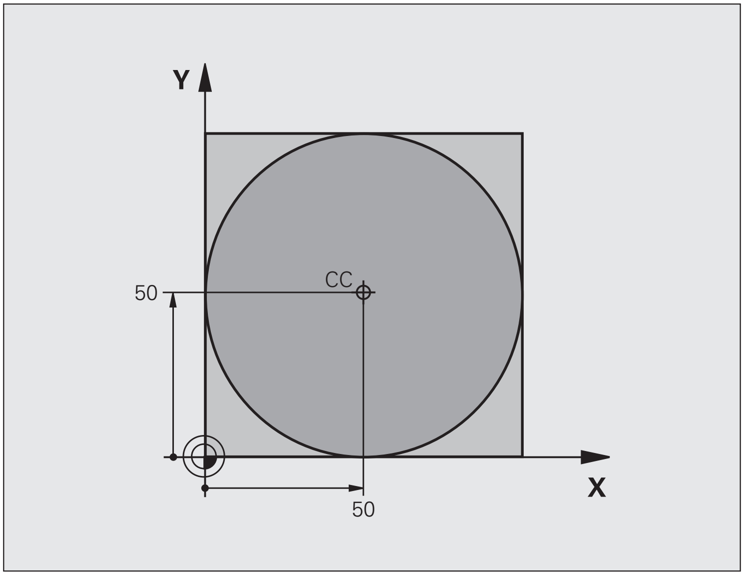
Example: Circular movements with Cartesian coordinates



%CIRCULAR G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define blank form for graphic workpiece simulation
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 T1 G17 S4000 *	Call tool in the spindle axis and with the spindle speed S
N40 G00 G40 G90 Z+250 *	Retract tool in the spindle axis at rapid traverse
N50 X-10 Y-10 *	Pre-position the tool
N60 G01 Z-5 F1000 M3 *	Move to working depth at feed rate F = 1000 mm/min
N70 G01 G41 X+5 Y+5 F300 *	Approach the contour at point 1, activate radius compensation G41
N80 G26 R5 F150 *	Tangential approach
N90 Y+85 *	Point 2: first straight line for corner 2
N100 G25 R10 *	Insert radius with R = 10 mm, feed rate: 150 mm/min
N110 X+30 *	Move to point 3: Starting point of the arc
N120 G02 X+70 Y+95 R+30 *	Move to point 4: end point of the arc with G02, radius 30 mm
N130 G01 X+95 *	Move to point 5
N140 Y+40 *	Move to point 6
N150 G06 X+40 Y+5 *	Move to point 7: End point of the arc, circular arc with tangential
	connection to point 6, TNC automatically calculates the radius

N160 G01 X+5 *	Move to last contour point 1
N170 G27 R5 F500 *	Depart the contour on a circular arc with tangential connection
N180 G40 X-20 Y-20 F1000 *	Retract tool in the working plane, cancel radius compensation
N190 G00 Z+250 M2 *	Retract tool in the tool axis, end of program
N99999999 %CIRCULAR G71 *	

Example: Full circle with Cartesian coordinates



`%C-CC G71 *`	
`N10 G30 G17 X+0 Y+0 Z-20 *`	Definition of workpiece blank
`N20 G31 G90 X+100 Y+100 Z+0 *`	
`N30 T1 G17 S3150 *`	Tool call
`N40 G00 G40 G90 Z+250 *`	Retract the tool
`N50 I+50 J+50 *`	Define the circle center
`N60 X-40 Y+50 *`	Pre-position the tool
`N70 G01 Z-5 F1000 M3 *`	Move to working depth
`N80 G41 X+0 Y+50 F300 *`	Approach starting point, radius compensation G41
`N90 G26 R5 F150 *`	Tangential approach
`N100 G02 X+0 *`	Move to the circle end point (= circle starting point)
`N110 G27 R5 F500 *`	Tangential exit
`N120 G01 G40 X-40 Y-50 F1000 *`	Retract tool in the working plane, cancel radius compensation
`N130 G00 Z+250 M2 *`	Retract tool in the tool axis, end of program
`N99999999 %C-CC G71 *`	

6.5 Path Contours—Polar Coordinates

Overview

With polar coordinates you can define a position in terms of its angle **H** and its distance **R** relative to a previously defined pole **I, J**.

Polar coordinates are useful with:

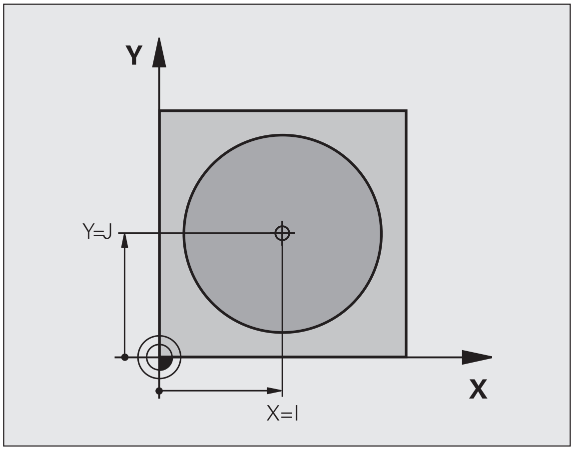
- Positions on circular arcs
- Workpiece drawing dimensions in degrees, e.g. bolt hole circles

Overview of path functions with polar coordinates

Function	Path function key	Tool movement	Required input	Page
Straight line **G10, G11**	L + P	Straight line	Polar radius, polar angle of the straight-line end point	Page 177
Circular arc **G12, G13**	C + P	Circular path around circle center/pole to arc end point	Polar angle of the arc end point,	Page 178
Circular arc **G15**	CR + P	Circular path corresponding to active direction of rotation	Polar angle of the circle end point	Page 178
Circular arc **G16**	CT + P	Circular arc with tangential connection to the preceding contour element	Polar radius, polar angle of the arc end point	Page 179
Helical interpolation	C + P	Combination of a circular and a linear movement	Polar radius, polar angle of the arc end point, coordinate of the end point in the tool axis	Page 180

Zero point for polar coordinates: pole I, J

You can define the pole CC anywhere in the part program before blocks containing polar coordinates. Set the pole in the same way as you would program the circle center.



SPEC FCT

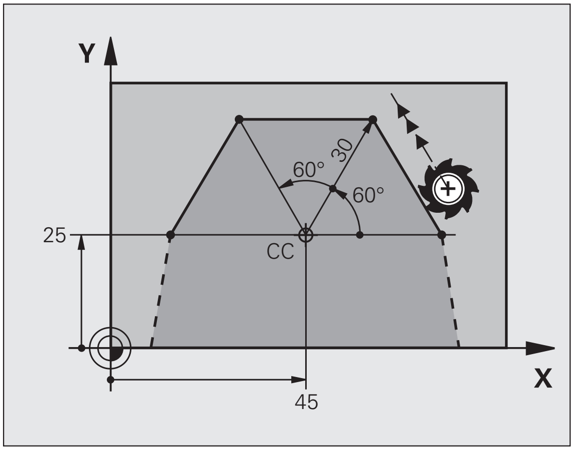
- To program a pole, press the SPEC FCT key
- Press the PROGRAM FUNCTIONS soft key
- Press the DIN/ISO soft key
- Press the I or J soft key
- **Coordinates**: Enter Cartesian coordinates for the pole or, if you want to use the last programmed position, enter **G29**. Before programming polar coordinates, define the pole. You can only define the pole in Cartesian coordinates. The pole remains in effect until you define a new pole

Example NC blocks

```
N120 I+45 J+45 *
```

Straight line at rapid traverse G10
Straight line with feed rate G11 F

The tool moves in a straight line from its current position to the straight-line end point. The starting point is the end point of the preceding block.



P

- **Polar coordinate radius R**: Enter the distance from the pole CC to the straight-line end point
- **Polar coordinate angle H**: Angular position of the straight-line end point between –360° and +360°

The sign of **H** depends on the angle reference axis:

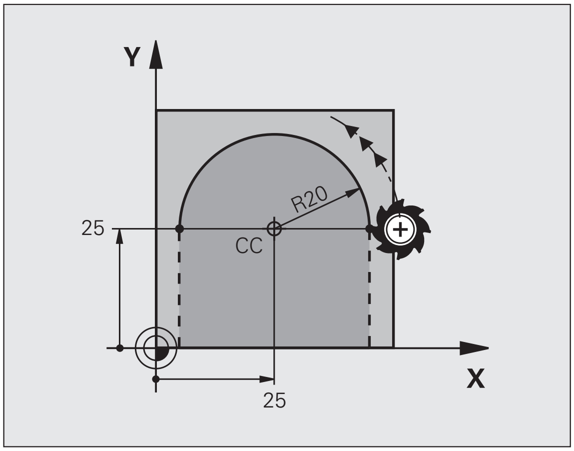
- If the angle from the angle reference axis to **R** is counterclockwise: **H**>0
- If the angle from the angle reference axis to **R** is clockwise: **H**<0

Example NC blocks

```
N120 I+45 J+45 *
N130 G11 G42 R+30 H+0 F300 M3 *
N140 H+60 *
N150 G91 H+60 *
N160 G90 H+180 *
```

Circular path G12/G13/G15 around pole I, J

The polar coordinate radius **R** is also the radius of the arc. **R** is defined by the distance from the starting point to the pole **I, J**. The last programmed tool position will be the starting point of the arc.



Direction of rotation

- In clockwise direction: **G12**
- In counterclockwise direction: **G13**
- Without programmed direction: **G15.** The TNC traverses the circular arc with the last programmed direction of rotation

P

- **Polar-coordinates angle H**: Angular position of the arc end point between –99 999.9999° and +99 999.9999°
- **Direction of rotation DR**

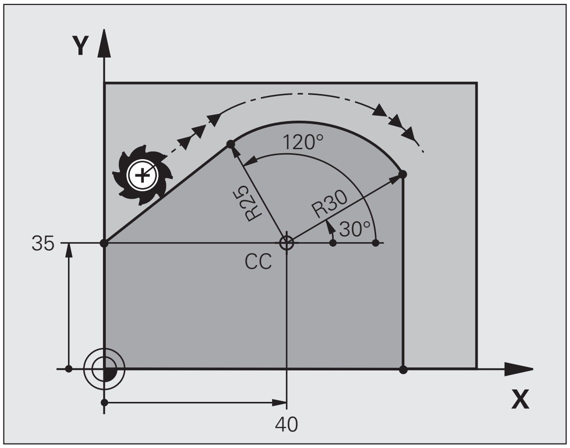
Example NC blocks

```
N180 I+25 J+25 *
N190 G11 G42 R+20 H+0 F250 M3 *
N200 G13 H+180 *
```

Circular path G16 with tangential connection

The tool moves on a circular path, starting tangentially from a preceding contour element.

- **Polar coordinate radius R**: Enter the distance from are end point to the pole **I, J**
- **Polar coordinates angle H**: Angular position of the arc end point



Example NC blocks

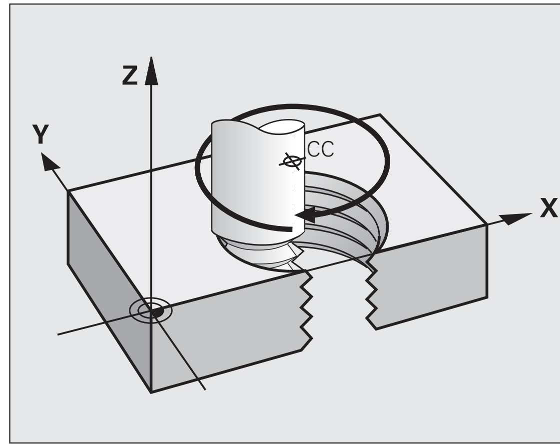
```
N120 I+40 J+35 *
N130 G01 G42 X+0 Y+35 F250 M3 *
N140 G11 R+25 H+120 *
N150 G16 R+30 H+30 *
N160 G01 Y+0 *
```

The pole is **not** the center of the contour arc!

Helical interpolation

A helix is a combination of a circular movement in a main plane and a linear movement perpendicular to this plane. You program the circular path in a main plane.

A helix is programmed only in polar coordinates.



Application

- Large-diameter internal and external threads
- Lubrication grooves

Calculating the helix

To program a helix, you must enter the total angle through which the tool is to move on the helix in incremental dimensions, and the total height of the helix.

For calculating a helix that is to be cut in an upward direction, you need the following data:

Thread revolutions n	Thread revolutions + thread overrun at thread beginning and end
Total height h	Thread pitch P times thread revolutions n
Incremental total angle **H**	Number of revolutions times 360° + angle for beginning of thread + angle for thread overrun
Starting coordinate Z	Pitch P times (thread revolutions + thread overrun at start of thread)

Shape of the helix

The table below illustrates in which way the shape of the helix is determined by the work direction, direction of rotation and radius compensation.

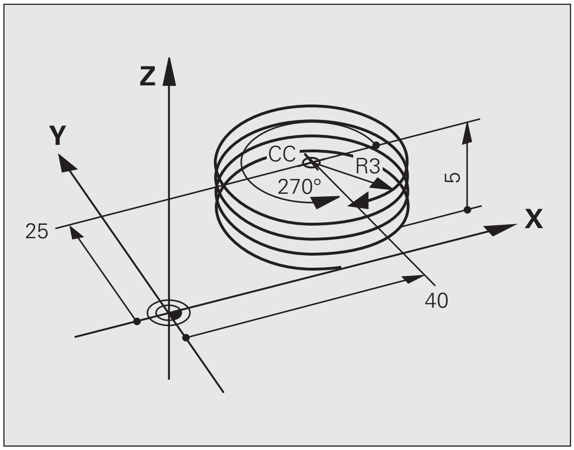
Internal thread	Work direction	Direction of rotation	Radius comp.
Right-handed Left-handed	Z+ Z+	**G13** **G12**	**G41** **G42**
Right-handed Left-handed	Z– Z–	**G12** **G13**	**G42** **G41**

External thread			
Right-handed Left-handed	Z+ Z+	**G13** **G12**	**G42** **G41**
Right-handed Left-handed	Z– Z–	**G12** **G13**	**G41** **G42**

Programming a helix

Always enter the same algebraic sign for the direction of rotation and the incremental total angle **G91 H**. The tool may otherwise move in a wrong path and damage the contour.

For the total angle **G91 H** you can enter a value of -99 999.9999° to +99 999.9999°.



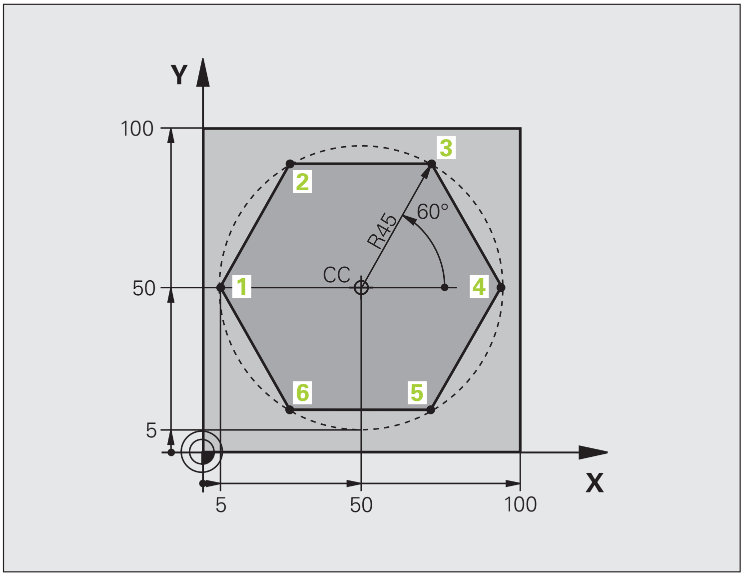
G 12

- **Polar coordinates angle:** Enter the total angle of tool traverse along the helix in incremental dimensions. **After entering the angle, specify the tool axis with an axis selection key.**
- **Coordinate:** Enter the coordinate for the height of the helix in incremental dimensions.
- Enter the **radius compensation** according to the table above

Example NC blocks: Thread M6 x 1 mm with 5 revolutions

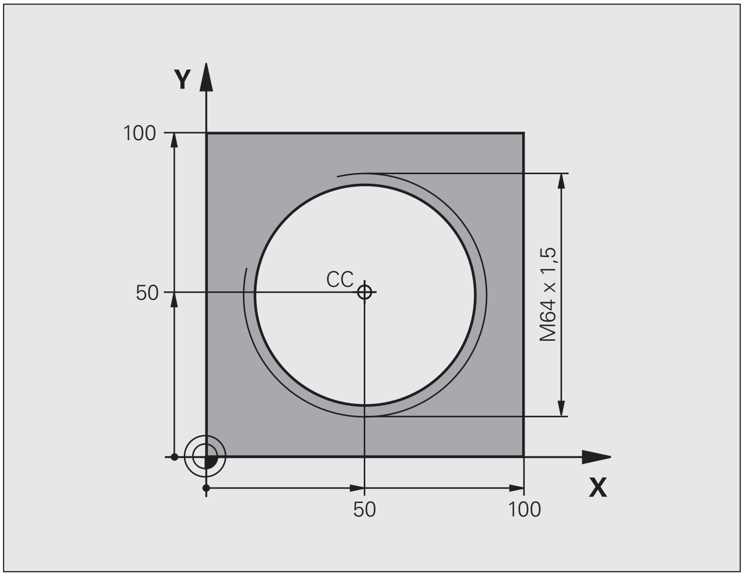
```
N120 I+40 J+25 *
N130 G01 Z+0 F100 M3 *
N140 G11 G41 R+3 H+270 *
N150 G12 G91 H-1800 Z+5 *
```

Example: Linear movement with polar coordinates

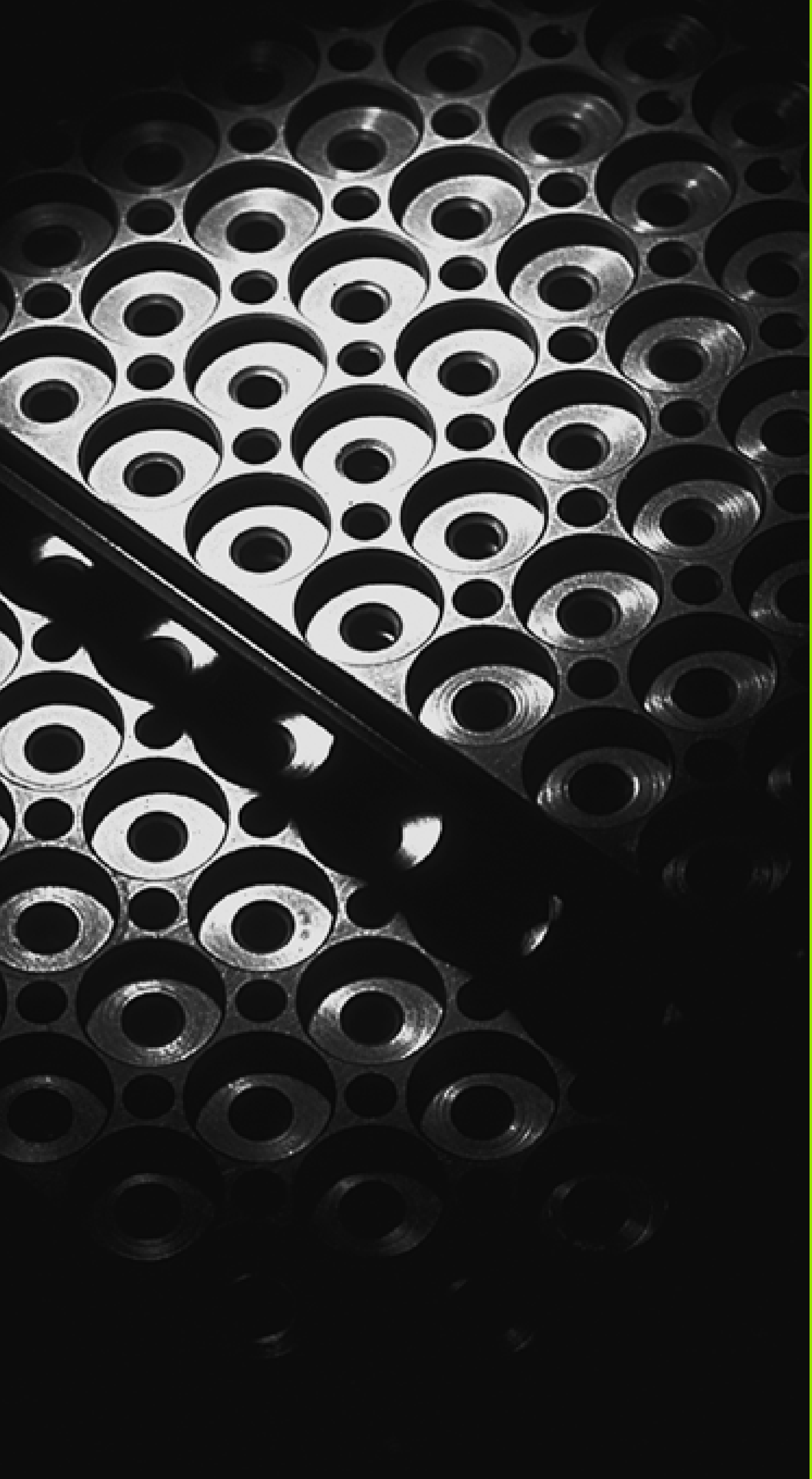


%LINEARPO G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Definition of workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 T1 G17 S4000 *	Tool call
N40 G00 G40 G90 Z+250 *	Define the datum for polar coordinates
N50 I+50 J+50 *	Retract the tool
N60 G10 R+60 H+180 *	Pre-position the tool
N70 G01 Z-5 F1000 M3 *	Move to working depth
N80 G11 G41 R+45 H+180 F250 *	Approach the contour at point 1
N90 G26 R5 *	Approach the contour at point 1
N100 H+120 *	Move to point 2
N110 H+60 *	Move to point 3
N120 H+0 *	Move to point 4
N130 H-60 *	Move to point 5
N140 H-120 *	Move to point 6
N150 H+180 *	Move to point 1
N160 G27 R5 F500 *	Tangential exit
N170 G40 R+60 H+180 F1000 *	Retract tool in the working plane, cancel radius compensation
N180 G00 Z+250 M2 *	Retract in the spindle axis, end of program
N99999999 %LINEARPO G71 *	

Example: Helix



%HELIX G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Definition of workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 T1 G17 S1400 *	Tool call
N40 G00 G40 G90 Z+250 *	Retract the tool
N50 X+50 Y+50 *	Pre-position the tool
N60 G29 *	Transfer the last programmed position as the pole
N70 G01 Z-12.75 F1000 M3 *	Move to working depth
N80 G11 G41 R+32 H+180 F250 *	Approach first contour point
N90 G26 R2 *	Connection
N100 G13 G91 H+3240 Z+13.5 F200 *	Helical interpolation
N110 G27 R2 F500 *	Tangential exit
N120 G01 G40 G90 X+50 Y+50 F1000 *	Retract in the tool axis, end program
N130 G00 Z+250 M2 *	



7

Programming: Subprograms and Program Section Repeats

7.1 Labeling Subprograms and Program Section Repeats

Subprograms and program section repeats enable you to program a machining sequence once and then run it as often as necessary.

Labels

The beginnings of subprograms and program section repeats are marked in a part program by labels **(G98 L)**.

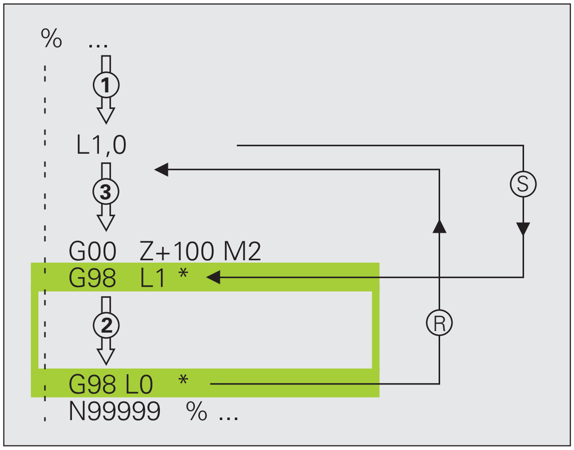
A LABEL is identified by a number between 1 and 999 or by a name you define. Each LABEL number or LABEL name can be set only once in the program with the LABEL SET key or by entering **G98**. The number of label names you can enter is only limited by the internal memory.

> Do not use a label number or label name more than once!

Label 0 (**G98 L0**) is used exclusively to mark the end of a subprogram and can therefore be used as often as desired.

7.2 Subprograms

Operating sequence



1. The TNC executes the part program up to the block in which a subprogram is called with **`Ln,0`**
2. The subprogram is then executed from beginning to end. The subprogram end is marked **`G98 L0`**
3. The TNC then resumes the part program from the block after the subprogram call **`Ln,0`**

Programming notes

- A main program can contain up to 254 subprograms
- You can call subprograms in any sequence and as often as desired
- A subprogram cannot call itself
- Write subprograms at the end of the main program (behind the block with M2 or M30)
- If subprograms are located before the block with M2 or M30, they will be executed at least once even if they are not called

Programming a subprogram

LBL SET

- ▶ To mark the beginning, press the LBL SET key
- ▶ Enter the subprogram number. If you want to use a label name, press the LBL NAME soft key to switch to text entry
- ▶ To mark the end, press the LBL SET key and enter the label number "0"

Calling a subprogram

LBL CALL

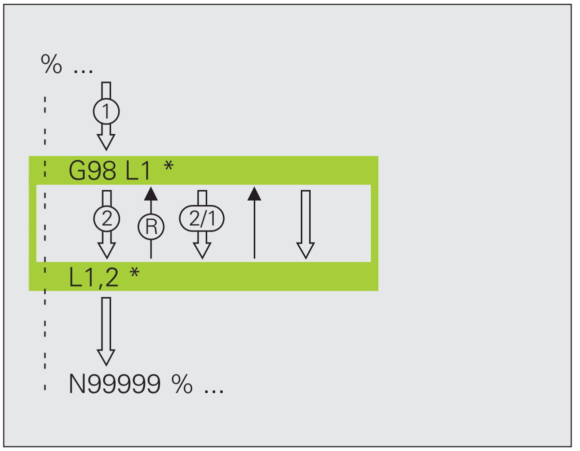
- ▶ To call a subprogram, press the LBL CALL key
- ▶ **`Label number`**: Enter the label number of the subprogram you wish to call. If you want to use a label name, press the LBL NAME soft key to switch to text entry

`G98 L 0` is not permitted (Label 0 is only used to mark the end of a subprogram).

7.3 Program Section Repeats

Label G98

The beginning of a program section repeat is marked by the label **G98 L**. The end of a program section repeat is identified by **Ln,m**.



Operating sequence

1. The TNC executes the part program up to the end of the program section (**Ln,m**)
2. Then the program section between the called LBL **Ln,m** is repeated the number of times entered for **m**
3. The TNC then resumes the part program after the last repetition

Programming notes

- You can repeat a program section up to 65 534 times in succession
- The total number of times the program section is executed is always one more than the programmed number of repeats

Programming a program section repeat

LBL SET

- To mark the beginning, press the LBL SET key and enter a LABEL NUMBER for the program section you wish to repeat. If you want to use a label name, press the LBL NAME soft key to switch to text entry
- Enter the program section

Calling a program section repeat

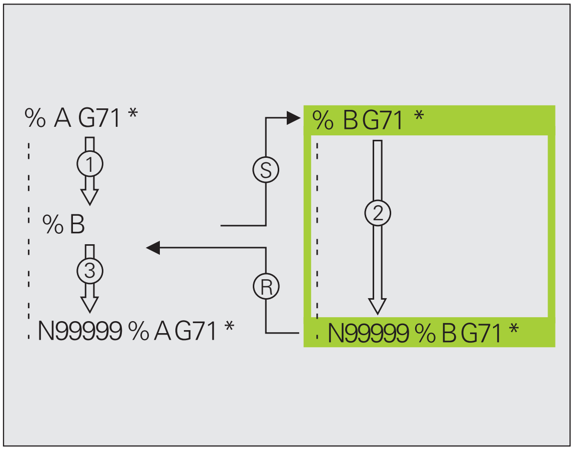
LBL CALL

- Press the LBL CALL key
- **To call subprograms/section repeats**: Enter the label number of the subprogram to be called, then confirm with the ENT key. If you want to use a label name, press the " key to switch to text entry
- **Repeat REP**: Enter the number of repeats, then confirm with the ENT key

7.4 Separate Program as Subprogram

Operating sequence

1 The TNC executes the part program up to the block in which another program is called with %
2 Then the other program is run from beginning to end
3 The TNC then resumes the first (calling) part program with the block after the program call



Programming notes

- No labels are needed to call any program as a subprogram
- The called program must not contain the miscellaneous functions M2 or M30. If you have defined subprograms with labels in the called program, you can then use M2 or M30 with the **D09 P01 +0 P02 +0 P03 99** jump function to force a jump over this program section
- The called program must not contain a % call into the calling program, otherwise an infinite loop will result

Calling any program as a subprogram

PGM CALL

- To select the functions for program call, press the PGM CALL key

PROGRAM

- Press the PROGRAM soft key for the TNC to start the dialog for defining the program to be called. Use the screen keyboard to enter the path name (GOTO key), or

SELECT PROGRAM

- press the SELECT PROGRAM soft key for the TNC to display a selection window in which you can select the program to be called. Confirm with the END key.

If the program you want to call is located in the same directory as the program you are calling it from, then you only need to enter the program name.

If the called program is not located in the same directory as the program you are calling it from, you must enter the complete path, e.g. **`TNC:\ZW35\SCHRUPP\PGM1.H`**

If you want to call a DIN/ISO program, enter the file type .I after the program name.

You can also call a program with Cycle **`G39`**.

As a rule, Q parameters are effective globally with a `%`. So please note that changes to Q parameters in the called program can also influence the calling program.

7.5 Nesting

Types of nesting

- Subprograms within a subprogram
- Program section repeats within a program section repeat
- Subprograms repeated
- Program section repeats within a subprogram

Nesting depth

The nesting depth is the number of successive levels in which program sections or subprograms can call further program sections or subprograms.

- Maximum nesting depth for subprograms: 8
- Maximum nesting depth for main program calls: 6, where a **G79** acts like a main program call
- You can nest program section repeats as often as desired

Subprogram within a subprogram

Example NC blocks

%SUBPGMS G71 *	
...	
N17 L "SP1",0 *	Subprogram at label G98 L SP1 is called
...	
N35 G00 G40 Z+100 M2 *	Last program block of the
	main program (with M2)
N36 G98 L "SP1"	Beginning of subprogram SP1
...	
N39 L2,0 *	Subprogram at label G98 L2 is called
...	
N45 G98 L0 *	End of subprogram 1
N46 G98 L2 *	Beginning of subprogram 2
...	
N62 G98 L0 *	End of subprogram 2
N99999999 %SUBPGMS G71 *	

Program execution

1. Main program SUBPGMS is executed up to block 17
2. Subprogram SP1 is called, and executed up to block 39
3. Subprogram 2 is called, and executed up to block 62. End of subprogram 2 and return jump to the subprogram from which it was called
4. Subprogram 1 is executed from block 40 up to block 45. End of subprogram 1 and return jump to the main program SUBPGMS
5. Main program SUBPGMS is executed from block 18 up to block 35. Return jump to block 1 and end of program

Repeating program section repeats

Example NC blocks

`%REPS G71 *`	
`...`	
`N15 G98 L1 *`	Beginning of program section repeat 1
`...`	
`N20 G98 L2 *`	Beginning of program section repeat 2
`...`	
`N27 L2,2 *`	Program section between G98 L2 (block N20)
`...`	and this block is repeated twice
`N35 L1,1 *`	Program section between G98 L1 (block N15)
`...`	and this block is repeated once
`N99999999 %REPS G71 *`	

Program execution

1. Main program REPS is executed up to block 27
2. Program section between block 20 and block 27 is repeated twice
3. Main program REPS is executed from block 28 to block 35
4. Program section between block 15 and block 35 is repeated once (including the program section repeat between 20 and block 27)
5. Main program REPS is executed from block 36 to block 50 (end of program)

Repeating a subprogram

Example NC blocks

`%SUBPGREP G71 *`	
`...`	
`N10 G98 L1 *`	Beginning of program section repeat 1
`N11 L2,0 *`	Subprogram call
`N12 L1,2 *`	Program section between G98 L1 (block N10)
`...`	and this block is repeated twice
`N19 G00 G40 Z+100 M2 *`	Last block of the main program with M2
`N20 G98 L2 *`	Beginning of subprogram
`...`	
`N28 G98 L0 *`	End of subprogram
`N99999999 %SUBPGREP G71 *`	

Program execution

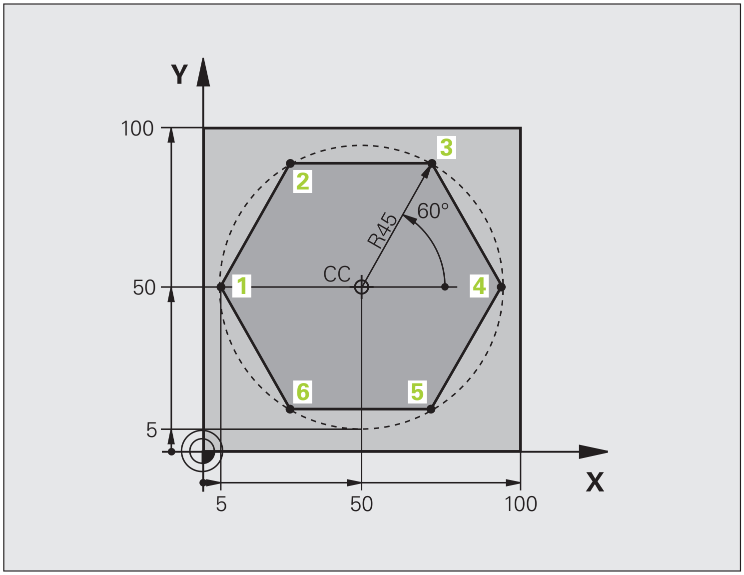
1. Main program SUBPGREP is executed up to block 11.
2. Subprogram 2 is called and executed.
3. Program section between block 10 and block 12 is repeated twice. Subprogram 2 is repeated twice.
4. Main program SUBPGREP is executed from block 13 to block 19. End of program.

7.6 Programming Examples

Example: Milling a contour in several infeeds

Program sequence

- Pre-position the tool to the workpiece surface
- Enter the infeed depth in incremental values
- Contour milling
- Repeat infeed and contour-milling



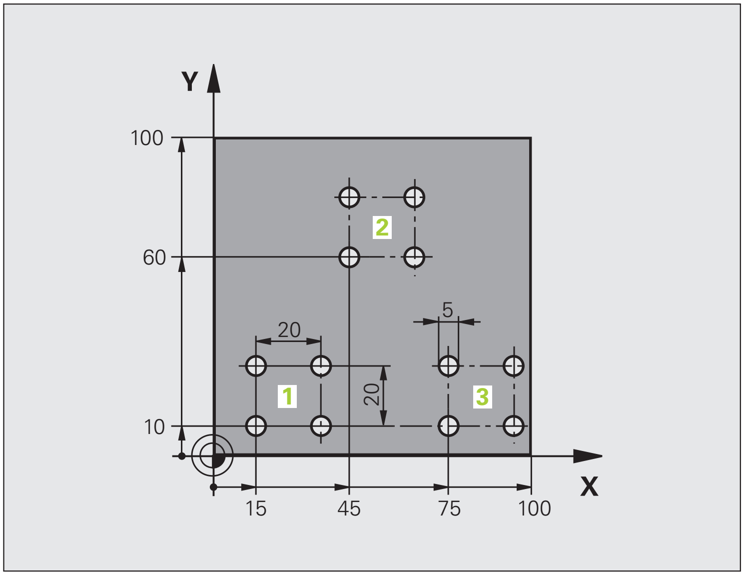
`%PGMWDH G71 *`	
`N10 G30 G17 X+0 Y+0 Z-40 *`	
`N20 G31 G90 X+100 Y+100 Z+0 *`	
`N30 T1 G17 S3500 *`	Tool call
`N40 G00 G40 G90 Z+250 *`	Retract the tool
`N50 I+50 J+50 *`	Set pole
`N60 G10 R+60 H+180 *`	Pre-position in the working plane
`N70 G01 Z+0 F1000 M3 *`	Pre-position to the workpiece surface

N80 G98 L1 *	Set label for program section repeat
N90 G91 Z-4 *	Infeed depth in incremental values (in space)
N100 G11 G41 G90 R+45 H+180 F250 *	First contour point
N110 G26 R5 *	Contour approach
N120 H+120 *	
N130 H+60 *	
N140 H+0 *	
N150 H-60 *	
N160 H-120 *	
N170 H+180 *	
N180 G27 R5 F500 *	Depart the contour
N190 G40 R+60 H+180 F1000 *	Retract tool
N200 L1,4 *	Return jump to label 1; section is repeated a total of 4 times
N200 G00 Z+250 M2 *	Retract in the tool axis, end program
N99999999 %PGMREP G71 *	

Example: Groups of holes

Program sequence

- Approach the groups of holes in the main program
- Call the group of holes (subprogram 1)
- Program the group of holes only once in subprogram 1



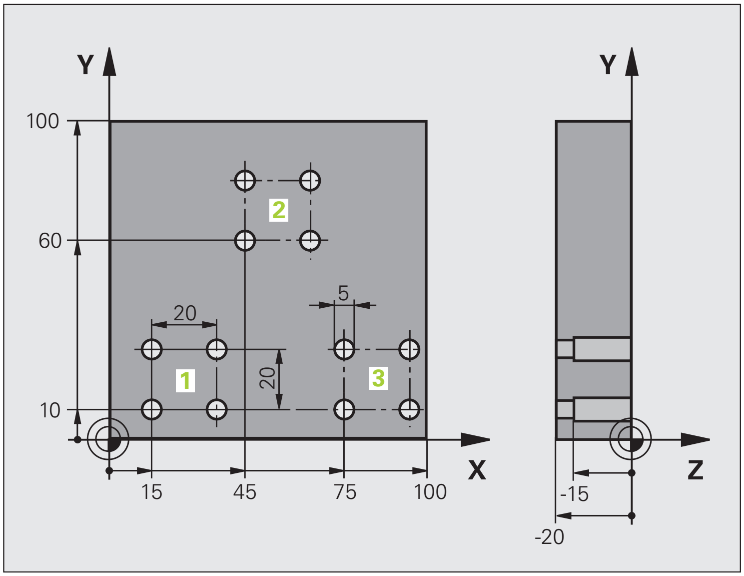
`%UP1 G71 *`	
`N10 G30 G17 X+0 Y+0 Z-40 *`	
`N20 G31 G90 X+100 Y+100 Z+0 *`	
`N30 T1 G17 S3500 *`	Tool call
`N40 G00 G40 G90 Z+250 *`	Retract the tool
`N50 G200 DRILLING`	Cycle definition: drilling
`Q200=2 ;SET-UP CLEARANCE`	
`Q201=-30 ;DEPTH`	
`Q206=300 ;FEED RATE FOR PLNGNG`	
`Q202=5 ;PLUNGING DEPTH`	
`Q210=0 ;DWELL TIME AT TOP`	
`Q203=+0 ;SURFACE COORDINATE`	
`Q204=2 ;2ND SET-UP CLEARANCE`	
`Q211=0 ;DWELL TIME AT DEPTH`	

N60 X+15 Y+10 M3 *	Move to starting point for group 1
N70 L1,0 *	Call the subprogram for the group
N80 X+45 Y+60 *	Move to starting point for group 2
N90 L1,0 *	Call the subprogram for the group
N100 X+75 Y+10 *	Move to starting point for group 3
N110 L1,0 *	Call the subprogram for the group
N120 G00 Z+250 M2 *	End of main program
N130 G98 L1 *	Beginning of subprogram 1: Group of holes
N140 G79 *	Call cycle for 1st hole
N150 G91 X+20 M99 *	Move to 2nd hole, call cycle
N160 Y+20 M99 *	Move to 3rd hole, call cycle
N170 X-20 G90 M99 *	Move to 4th hole, call cycle
N180 G98 L0 *	End of subprogram 1
N99999999 %UP1 G71 *	

Example: Group of holes with several tools

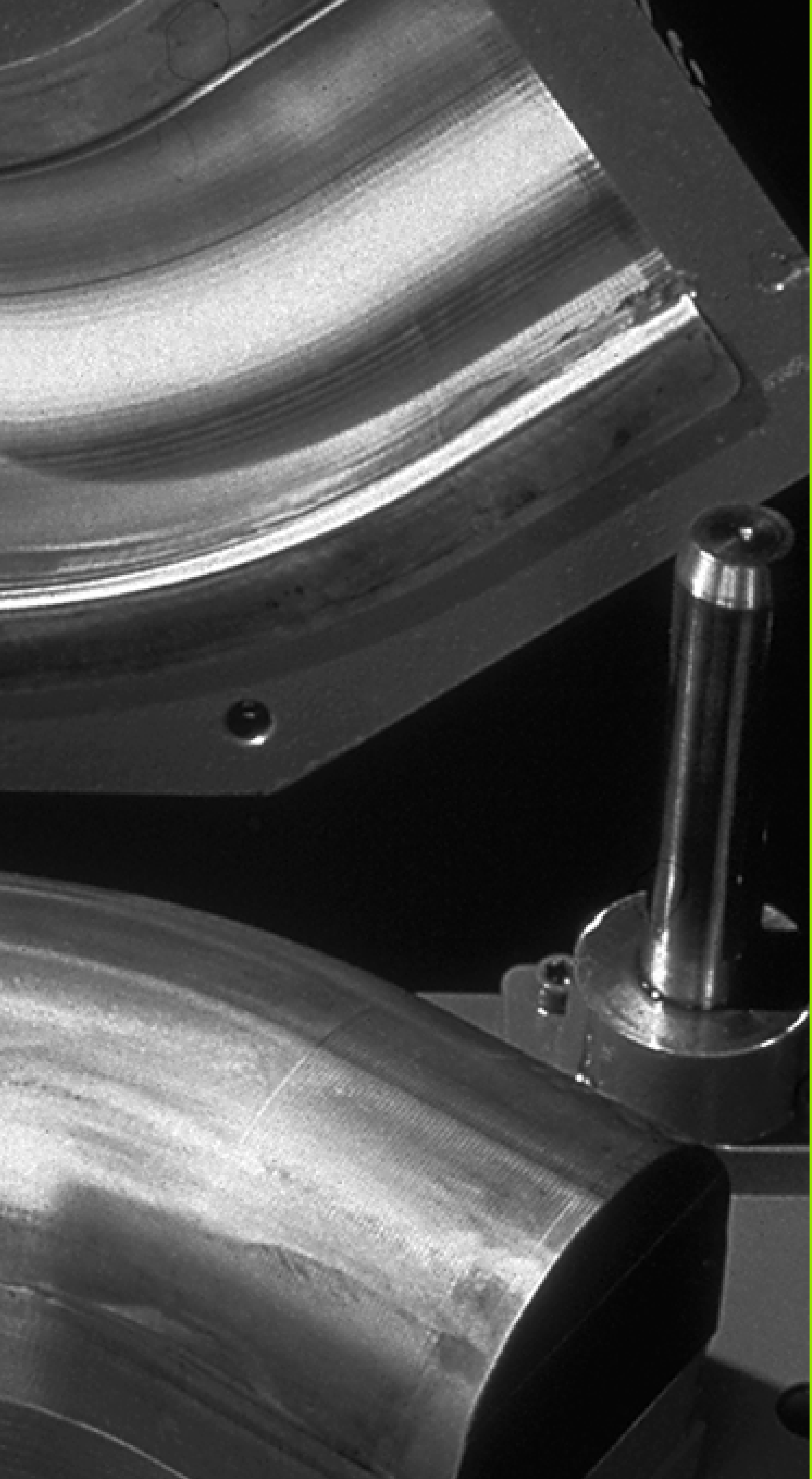
Program sequence

- Program the fixed cycles in the main program
- Call the entire hole pattern (subprogram 1)
- Approach the groups of holes in subprogram 1, call group of holes (subprogram 2)
- Program the group of holes only once in subprogram 2



`%SP2 G71 *`	
`N10 G30 G17 X+0 Y+0 Z-40 *`	
`N20 G31 G90 X+100 Y+100 Z+0 *`	
`N30 T1 G17 S5000 *`	Call tool: center drill
`N40 G00 G40 G90 Z+250 *`	Retract the tool
`N50 G200 DRILLING`	Cycle definition: CENTERING
`Q200=2 ;SET-UP CLEARANCE`	
`Q201=-3 ;DEPTH`	
`Q206=250 ;FEED RATE FOR PLNGNG`	
`Q202=3 ;PLUNGING DEPTH`	
`Q210=0 ;DWELL TIME AT TOP`	
`Q203=+0 ;SURFACE COORDINATE`	
`Q204=10 ;2ND SET-UP CLEARANCE`	
`Q211=0.2 ;DWELL TIME AT DEPTH`	
`N60 L1,0 *`	Call subprogram 1 for the entire hole pattern

N70 G00 Z+250 M6 *	Tool change
N80 T2 G17 S4000 *	Call tool: drill
N90 D0 Q201 P01 -25 *	New depth for drilling
N100 D0 Q202 P01 +5 *	New plunging depth for drilling
N110 L1,0 *	Call subprogram 1 for the entire hole pattern
N120 G00 Z+250 M6 *	Tool change
N130 T3 G17 S500 *	Call tool: reamer
N140 G201 REAMING	Cycle definition: REAMING
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q211=0.5 ;DWELL TIME AT DEPTH	
Q208=400 ;RETRACTION FEED RATE	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
N150 L1,0 *	Call subprogram 1 for the entire hole pattern
N160 G00 Z+250 M2 *	End of main program
N170 G98 L1 *	Beginning of subprogram 1: Entire hole pattern
N180 G00 G40 G90 X+15 Y+10 M3 *	Move to starting point for group 1
N190 L2,0 *	Call subprogram 2 for the group
N200 X+45 Y+60 *	Move to starting point for group 2
N210 L2,0 *	Call subprogram 2 for the group
N220 X+75 Y+10 *	Move to starting point for group 3
N230 L2,0 *	Call subprogram 2 for the group
N240 G98 L0 *	End of subprogram 1
N250 G98 L2 *	Beginning of subprogram 2: Group of holes
N260 G79 *	Call cycle for 1st hole
N270 G91 X+20 M99 *	Move to 2nd hole, call cycle
N280 Y+20 M99 *	Move to 3rd hole, call cycle
N290 X-20 G90 M99 *	Move to 4th hole, call cycle
N300 G98 L0 *	End of subprogram 2
N310 %SP2 G71 *	

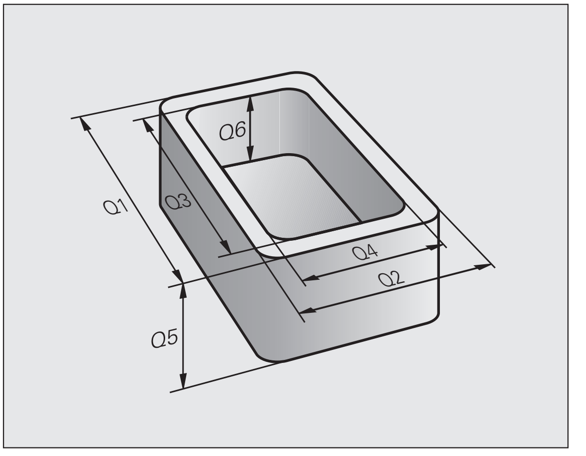


8

Programming: Q Parameters

8.1 Principle and Overview

You can program entire families of parts in a single part program. You do this by entering variables called Q parameters instead of fixed numerical values.



Q parameters can represent information such as:

- Coordinate values
- Feed rates
- Rotational speeds
- Cycle data

Q parameters also enable you to program contours that are defined with mathematical functions. You can also use Q parameters to make the execution of machining steps depend on logical conditions.

Q parameters are designated by letters and a number between 0 and 1999. Parameters that take effect in different manners are available. Please refer to the following table:

Meaning	Range
Freely applicable parameters, as long as no overlapping with SL cycles can occur. They are globally effective for all programs stored in the TNC memory.	**Q0** to **Q99**
Parameters for special TNC functions	**Q100** to **Q199**
Parameters that are primarily used for cycles; they are globally effective for all programs stored in the TNC memory	**Q200** to **Q1199**
Parameters that are primarily used for OEM cycles, and are globally effective for all programs stored in the TNC memory. This may require coordination with the machine manufacturer or supplier.	**Q1200** to **Q1399**
Parameters that are primarily used for **call-active** OEM cycles, and are globally effective for all programs that are stored in the TNC memory	**Q1400** to **Q1499**
Parameters that are primarily used for **Def-active** OEM cycles, and are globally effective for all programs that are stored in the TNC memory	**Q1500** to **Q1599**
Freely applicable parameters, globally effective for all programs stored in the TNC memory	**Q1600** to **Q1999**

Meaning	Range
Freely usable **QL** parameters, only effective locally (within a program)	**QL0** to **QL499**
Freely usable **QR** parameters that are nonvolatile, i.e. they **r**emain in effect even after a power interruption	**QR0** to **QR499**

QS parameters (the **S** stands for string) are also available on the TNC and enable you to process texts. In principle, the same ranges are available for **QS** parameters as for Q parameters (see table above).

Note that for the **QS** parameters the **QS100** to **QS199** range is reserved for internal texts.

Local parameters **QL** are only effective within the respective program, and are not applied as part of program calls or macros.

Programming notes

You can mix Q parameters and fixed numerical values within a program.

Q parameters can be assigned numerical values between -999 999 999 and +999 999 999. The input range is limited to 15 digits, of which 9 may be before the decimal point. Internally the TNC calculates numbers up to a value of 10^{10}.

You can assign a maximum of 254 characters to **QS** parameters.

Some Q and QS parameters are always assigned the same data by the TNC. For example, **Q108** is always assigned the current tool radius (see "Preassigned Q Parameters", page 257).

Calling Q-parameter functions

When you are writing a part program, press the “Q” key (in the numeric keypad for numerical input and axis selection, below the +/– key). The TNC then displays the following soft keys:

Function group	Soft key	Page
Basic arithmetic (assign, add, subtract, multiply, divide, square root)	BASIC ARITHM.	Page 206
Trigonometric functions	TRIGO- NOMETRY	Page 208
If/then conditions, jumps	JUMP	Page 210
Other functions	DIVERSE FUNCTION	Page 212
Entering formulas directly	FORMULA	Page 241
Function for machining complex contours	CONTOUR FORMULA	See User’s Manual for Cycles

The TNC shows the soft keys Q, QL and QR when you are defining or assigning a Q parameter. First press one of these soft keys to select the desired type of parameter, and then enter the parameter number.

If you have a USB keyboard connected, you can press the Q key to open the dialog for entering a formula.

8.2 Part Families—Q Parameters in Place of Numerical Values

Function

The Q parameter function **D0: ASSIGN** assigns numerical values to Q parameters. This enables you to use variables in the program instead of fixed numerical values.

Example NC blocks

`N150 D00 Q10 P01 +25 *`	Assignment
`...`	Q10 is assigned the value 25
`N250  G00 X +Q10 *`	Corresponds to G00 X +25

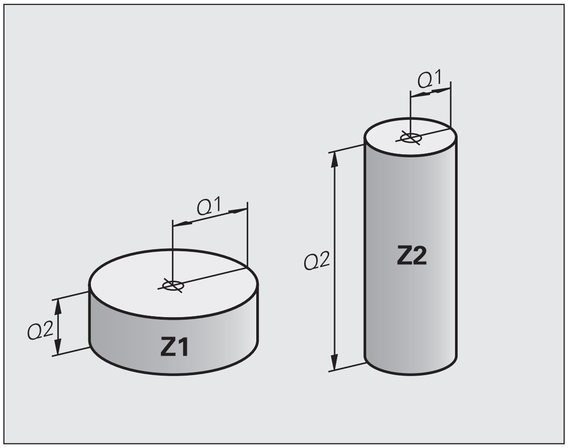
You need write only one program for a whole family of parts, entering the characteristic dimensions as Q parameters.

To program a particular part, you then assign the appropriate values to the individual Q parameters.

Example

Cylinder with Q parameters

Cylinder radius	R = Q1
Cylinder height	H = Q2
Cylinder Z1	Q1 = +30 Q2 = +10
Cylinder Z2	Q1 = +10 Q2 = +50



8.3 Describing Contours through Mathematical Operations

Function

The Q parameters listed below enable you to program basic mathematical functions in a part program:

- Select a Q-parameter function: Press the Q key (in the numerical keypad at right). The Q-parameter functions are displayed in a soft-key row
- To select the mathematical functions, press the BASIC ARITHMETIC soft key. The TNC then displays the following soft keys:

Overview

Function	Soft key
D00: ASSIGN Example: **D00 Q5 P01 +60 *** Assigns a numerical value.	D0 X = Y
D01: ADDITION Example: **D01 Q1 P01 -Q2 P02 -5 *** Calculates and assigns the sum of two values.	D1 X + Y
D02: SUBTRACTION Example: **D02 Q1 P01 +10 P02 +5 *** Calculates and assigns the difference of two values.	D2 X - Y
D03: MULTIPLICATION Example: **D03 Q2 P01 +3 P02 +3 *** Calculates and assigns the product of two values.	D3 X * Y
D04: DIVISION Example: **D04 Q4 P01 +8 P02 +Q2 *** Calculates and assigns the quotient of two values. **Not permitted:** Division by 0	D4 X / Y
D05: SQUARE ROOT Example: **D05 Q50 P01 4 *** Calculates and assigns the square root of a number. **Not permitted:** Calculating the square root of a negative value!	D5 SQRT

To the right of the "=" character you can enter the following:

- Two numbers
- Two Q parameters
- A number and a Q parameter

The Q parameters and numerical values in the equations can be entered with positive or negative signs.

Programming fundamental operations

Example:

Q	Call the Q parameter functions by pressing the Q key
BASIC ARITHM.	To select the mathematical functions, press the BASIC ARITHMETIC soft key
D0 X = Y	To select the Q parameter function ASSIGN, press the D0 X = Y soft key

PARAMETER NO. FOR RESULT?

5 ENT	Enter the number of the Q parameter, e.g. 5

1ST VALUE OR PARAMETER?

10 ENT	Assign the value 10 to Q5

Q	Call the Q parameter functions by pressing the Q key
BASIC ARITHM.	To select the mathematical functions, press the BASIC ARITHMETIC soft key
D3 X * Y	To select the Q parameter function MULTIPLICATION, press the D3 X * Y soft key

PARAMETER NO. FOR RESULT?

12 ENT	Enter the number of the Q parameter, e.g. 12

1ST VALUE OR PARAMETER?

Q5 ENT	Enter Q5 for the first value

2ND VALUE OR PARAMETER?

7 ENT	Enter 7 for the second value

Example: Program blocks in the TNC

```
N17 D00 Q5 P01 +10 *
N17 D03 Q12 P01 +Q5 P02 +7 *
```

8.4 Trigonometric Functions

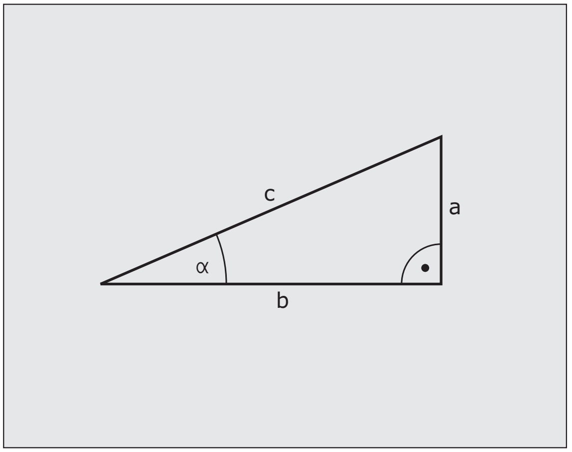
Definitions

Sine, cosine and tangent are terms designating the ratios of sides of right triangles. In this case:

Sine: $\sin \alpha = a / c$
Cosine: $\cos \alpha = b / c$
Tangent: $\tan \alpha = a / b = \sin \alpha / \cos \alpha$

where

- c is the side opposite the right angle
- a is the side opposite the angle α
- b is the third side.



The TNC can find the angle from the tangent:

$\alpha = \text{arc tan } (a / b) = \text{arc tan } (\sin \alpha / \cos \alpha)$

Example:

a = 25 mm

b = 50 mm

$\alpha = \text{arc tan } (a / b) = \text{arc tan } 0.5 = 26.57°$

Furthermore:

$a^2 + b^2 = c^2$ (where $a^2 = a \times a$)

$c = \sqrt{(a^2 + b^2)}$

Programming trigonometric functions

Press the ANGLE FUNCTION soft key to call the trigonometric functions. The TNC then displays the following soft keys:

Programming: Compare "Example: Programming fundamental operations."

Function	Soft key
D06: SINE Example: **`D06 Q20 P01 -Q5 *`** Calculates and assigns the sine of an angle in degrees (°)	D6 SIN(X)
D07: COSINE Example: **`D07 Q21 P01 -Q5 *`** Calculates and assigns the cosine of an angle in degrees (°)	FN7 COS(X)
D08: ROOT SUM OF SQUARES Example: **`D08 Q10 P01 +5 P02 +4 *`** Calculates and assigns length from two values	D8 X LEN Y
D13: ANGLE Example: **`D13 Q20 P01 +10 P02 -Q1 *`** Calculates the angle from the arc tangent of two sides or from the sine and cosine of the angle (°0 < angle < 360°) and assigns it to a parameter	D13 X ANG Y

8.5 If-Then Decisions with Q Parameters

Function

The TNC can make logical If-Then decisions by comparing a Q parameter with another Q parameter or with a numerical value. If the condition is fulfilled, the TNC continues the program at the label that is programmed after the condition (for information on labels, see "Labeling Subprograms and Program Section Repeats", page 186). If it is not fulfilled, the TNC continues with the next block.

To call another program as a subprogram, enter a % program call after the block with the target label.

Unconditional jumps

An unconditional jump is programmed by entering a conditional jump whose condition is always true. Example:

D09 P01 +10 P02 +10 P03 1 *

Programming If-Then decisions

Press the JUMP soft key to call the If-Then conditions. The TNC then displays the following soft keys:

Function	Soft key
D09: IF EQUAL, JUMP Example: **D09 P01 +Q1 P02 +Q3 P03 "SPCAN25" *** If the two values or parameters are equal, jump to the given label.	D9 IF X EQ Y GOTO
D10: IF UNEQUAL, JUMP Example: **D10 P01 +10 P02 -Q5 P03 10 *** If the two values or parameters are unequal, jump to the given label.	D10 IF X NE Y GOTO
D11: IF GREATER, JUMP Example: **D11 P01 +Q1 P02 +10 P03 5 *** If the first value or parameter is greater than the second, jump to the given label.	D11 IF X GT Y GOTO
D12: IF LESS, JUMP Example: **D12 P01 +Q5 P02 +0 P03 "ANYNAME" *** If the first value or parameter is less than the second, jump to the given label.	D12 IF X LT Y GOTO

8.6 Checking and Changing Q Parameters

Procedure

You can check Q parameters when writing, testing and running programs in all operating modes, and also edit them.

▶ If you are in a program run, interrupt it if required (for example, by pressing the machine STOP button and the INTERNAL STOP soft key). If you are in a test run, interrupt it.

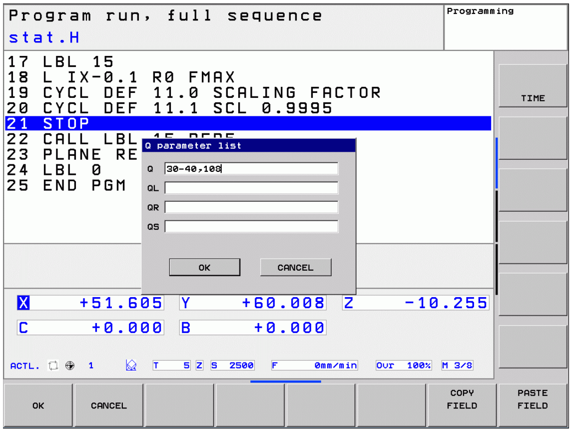
Q INFO

▶ To call Q parameter functions, press the Q INFO soft key or the Q key.

▶ The TNC lists all parameters and their current values. Use the arrow keys or the GOTO key to select the desired parameter.

▶ If you would like to change the value, press the EDIT CURRENT FIELD soft key, enter the new value, and confirm with the ENT key.

▶ To leave the value unchanged, press the PRESENT VALUE soft key or end the dialog with the END key.



> The parameters used internally or by the TNC in cycles are commented.
>
> If you want to check or edit local, global or string parameters, press the SHOW PARAMETERS Q QL QR QS soft key. The TNC then displays all respective parameters and the functions described above also apply.

You can have the Q parameters be shown in the additional status display in the Manual, Electronic Handwheel, Single Block, Full Sequence and Test Run operating modes.

▶ If you are in a program run, interrupt it if required (for example, by pressing the machine STOP button and the INTERNAL STOP soft key). If you are in a test run, interrupt it.

▶ Call the soft-key row for screen layout.

PROGRAM + STATUS

▶ Select the screen layout with additional status display: In the right half of the screen, the TNC shows the **Overview** status form.

STATUS OF Q PARAM.

▶ Press the STATUS OF Q PARAM. soft key.

Q PARAMETER LIST

▶ Press the Q PARAMETER LIST soft key

▶ The TNC opens a pop-up window in which you can enter the desired range for display of the Q parameters or string parameters. Multiple Q parameters are entered separated by commas (e.g. Q 1,2,3,4). To define display ranges, enter a hyphen (e.g. Q 10-14).

8.7 Additional Functions

Overview

Press the DIVERSE FUNCTION soft key to call the additional functions. The TNC then displays the following soft keys:

Function	Soft key	Page
D14:ERROR Output of error messages	D14 ERROR=	Page 213
D19:PLC Send values to the PLC	D19 PLC=	Page 227
D29:PLC Transfer up to eight values to the PLC	D29 PLC LIST=	Page 228
D37:EXPORT Export local Q parameters or QS parameters into a calling program	D37 EXPORT	Page 229

D14: ERROR: Displaying error messages

With the function **D14** you can call messages under program control. The messages are predefined by the machine tool builder or by HEIDENHAIN. Whenever the TNC comes to a block with **D14** in the Program Run or Test Run mode, it interrupts the program run and displays a message. The program must then be restarted from this point. The error numbers are listed in the table below.

Range of error numbers	Standard dialog text
0 ... 999	Machine-dependent dialog
1000 ... 1199	Internal error messages (see table)

Example NC block

The TNC is to display the text stored under error number 254:

```
N180 D14 P01 254 *
```

Error message predefined by HEIDENHAIN

Error number	Text
1000	Spindle?
1001	Tool axis is missing
1002	Tool radius too small
1003	Tool radius too large
1004	Range exceeded
1005	Start position incorrect
1006	ROTATION not permitted
1007	SCALING FACTOR not permitted
1008	MIRROR IMAGE not permitted
1009	Datum shift not permitted
1010	Feed rate is missing
1011	Entry value incorrect
1012	Incorrect sign
1013	Entered angle not permitted
1014	Touch point inaccessible
1015	Too many points
1016	Contradictory input
1017	CYCL incomplete

Error number	Text
1018	Plane wrongly defined
1019	Wrong axis programmed
1020	Wrong rpm
1021	Radius comp. undefined
1022	Rounding-off undefined
1023	Rounding radius too large
1024	Program start undefined
1025	Excessive nesting
1026	Angle reference missing
1027	No fixed cycle defined
1028	Slot width too small
1029	Pocket too small
1030	Q202 not defined
1031	Q205 not defined
1032	Q218 must be greater than Q219
1033	CYCL 210 not permitted
1034	CYCL 211 not permitted
1035	Q220 too large
1036	Q222 must be greater than Q223
1037	Q244 must be greater than 0
1038	Q245 must not equal Q246
1039	Angle range must be < 360°
1040	Q223 must be greater than Q222
1041	Q214: 0 not permitted

Error number	Text
1042	Traverse direction not defined
1043	No datum table active
1044	Position error: center in axis 1
1045	Position error: center in axis 2
1046	Hole diameter too small
1047	Hole diameter too large
1048	Stud diameter too small
1049	Stud diameter too large
1050	Pocket too small: rework axis 1
1051	Pocket too small: rework axis 2
1052	Pocket too large: scrap axis 1
1053	Pocket too large: scrap axis 2
1054	Stud too small: scrap axis 1
1055	Stud too small: scrap axis 2
1056	Stud too large: rework axis 1
1057	Stud too large: rework axis 2
1058	TCHPROBE 425: length exceeds max
1059	TCHPROBE 425: length below min
1060	TCHPROBE 426: length exceeds max
1061	TCHPROBE 426: length below min
1062	TCHPROBE 430: diameter too large
1063	TCHPROBE 430: diameter too small
1064	No measuring axis defined
1065	Tool breakage tolerance exceeded
1066	Enter Q247 unequal to 0
1067	Enter Q247 greater than 5
1068	Datum table?
1069	Enter Q351 unequal 0
1070	Thread depth too large

Error number	Text
1071	Missing calibration data
1072	Tolerance exceeded
1073	Block scan active
1074	ORIENTATION not permitted
1075	3-D ROT not permitted
1076	Activate 3-D ROT
1077	Enter a negative value for the depth
1078	Q303 not defined in measuring cycle
1079	Tool axis not allowed
1080	Calculated values incorrect
1081	Contradictory measuring points
1082	Clearance height entered incorrectly
1083	Contradictory type of plunging
1084	Machining cycle not permitted
1085	Line is write-protected
1086	Oversize greater than depth
1087	No point angle defined
1088	Contradictory data
1089	Slot position 0 not permitted
1090	Enter infeed unequal to 0
1091	Switchover of Q399 not allowed
1092	Tool not defined
1093	Tool number not permitted
1094	Tool name not allowed
1095	Software option not active
1096	Kinematics cannot be restored
1097	Function not permitted
1098	Contradictory workpc. blank dim.
1099	Measuring position not allowed

Error number	Text
1100	Kinematic access not possible
1101	Meas. pos. not in traverse range
1102	Preset compensation not possible
1103	Tool radius too large
1104	Plunging type is not possible
1105	Plunge angle incorrectly defined
1106	Angular length is undefined
1107	Slot width is too large
1108	Scaling factors not equal
1109	Tool data inconsistent

D18: Read system data

With the D18 function you can read system data and store them in Q parameters. You select the system data through a group name (ID number), and additionally through a number and an index.

Group name, ID no.	Number	Index	Meaning
Program information, 10	3	-	Number of the active fixed cycle
	103	Q parameter number	Relevant within NC cycles; for inquiry as to whether the Q parameter given under IDX was explicitly stated in the associated CYCLE DEF.
System jump addresses, 13	1	-	Label jumped to during M2/M30 instead of ending the current program. Value = 0: M2/M30 has the normal effect
	2	-	Label jumped to if FN14: ERROR after the NC CANCEL reaction instead of aborting the program with an error. The error number programmed in the FN14 command can be read under ID992 NR14. Value = 0: FN14 has the normal effect.
	3	-	Label jumped to in the event of an internal server error (SQL, PLC, CFG) instead of aborting the program with an error message. Value = 0: Server error has the normal effect.
Machine status, 20	1	-	Active tool number
	2	-	Prepared tool number
	3	-	Active tool axis 0=X, 1=Y, 2=Z, 6=U, 7=V, 8=W
	4	-	Programmed spindle speed
	5	-	Active spindle status: –1=undefined, 0=M3 active, 1=M4 active, 2=M5 after M3, 3=M5 after M4
	7	-	Gear range
	8	-	Coolant status: 0=off, 1=on
	9	-	Active feed rate
	10	-	Index of prepared tool
	11	-	Index of active tool
Channel data, 25	1	-	Channel number
Cycle parameter, 30	1	-	Set-up clearance of active fixed cycle
	2	-	Drilling depth / milling depth of active fixed cycle
	3	-	Plunging depth of active fixed cycle

Group name, ID no.	Number	Index	Meaning
	4	-	Feed rate for pecking in active fixed cycle
	5	-	1st side length for rectangular pocket cycle
	6	-	2nd side length for rectangular pocket cycle
	7	-	1st side length for slot cycle
	8	-	2nd side length for slot cycle
	9	-	Radius for circular pocket cycle
	10	-	Feed rate for milling in active fixed cycle
	11	-	Direction of rotation for active fixed cycle
	12	-	Dwell time for active fixed cycle
	13	-	Thread pitch for Cycles 17, 18
	14	-	Milling allowance for active fixed cycle
	15	-	Direction angle for rough out in active fixed cycle
	21	-	Probing angle
	22	-	Probing path
	23	-	Probing feed rate
Modal condition, 35	1	-	Dimensioning: 0 = absolute (G90) 1 = incremental (G91)
Data for SQL tables, 40	1	-	Result code for the last SQL command
Data from the tool table, 50	1	Tool no.	Tool length
	2	Tool no.	Tool radius
	3	Tool no.	Tool radius R2
	4	Tool no.	Tool length oversize DL
	5	Tool no.	Tool radius oversize DR
	6	Tool no.	Tool radius oversize DR2
	7	Tool no.	Tool inhibited (0 or 1)
	8	Tool no.	Number of the replacement tool
	9	Tool no.	Maximum tool age TIME1
	10	Tool no.	Maximum tool age TIME2
	11	Tool no.	Current tool age CUR. TIME

Group name, ID no.	Number	Index	Meaning
	12	Tool no.	PLC status
	13	Tool no.	Maximum tooth length LCUTS
	14	Tool no.	Maximum plunge angle ANGLE
	15	Tool no.	TT: Number of teeth CUT
	16	Tool no.	TT: Wear tolerance in length LTOL
	17	Tool no.	TT: Wear tolerance in radius RTOL
	18	Tool no.	TT: Rotational direction DIRECT (0=positive/-1=negative)
	19	Tool no.	TT: Offset in plane R-OFFS
	20	Tool no.	TT: Offset in length L-OFFS
	21	Tool no.	TT: Break tolerance for length LBREAK
	22	Tool no.	TT: Break tolerance for radius RBREAK
	23	Tool no.	PLC value
	24	Tool no.	Probe-center offset in reference axis CAL-OF1
	25	Tool no.	Probe-center offset in minor axis CAL-OF2
	26	Tool no.	Spindle angle for calibration CAL-ANG
	27	Tool no.	Tool type for pocket table
	28	Tool no.	Maximum rpm NMAX
Pocket table data, 51	1	Pocket number	Tool number
	2	Pocket number	Special tool: 0=no, 1=yes
	3	Pocket number	Fixed pocket: 0=no, 1=yes
	4	Pocket number	Locked pocket: 0=no, 1=yes
	5	Pocket number	PLC status
Pocket number of a tool in the tool-pocket table, 52	1	Tool no.	Pocket number
	2	Tool no.	Tool magazine number
Values programmed immediately after TOOL CALL, 60	1	-	Tool number T
	2	-	Active tool axis 0 = X 6 = U 1 = Y 7 = V 2 = Z 8 = W

Group name, ID no.	Number	Index	Meaning
	3	-	Spindle speed S
	4	-	Tool length oversize DL
	5	-	Tool radius oversize DR
	6	-	Automatic TOOL CALL 0 = yes, 1 = no
	7	-	Tool radius oversize DR2
	8	-	Tool index
	9	-	Active feed rate
Values programmed immediately after TOOL DEF, 61	1	-	Tool number T
	2	-	Length
	3	-	Radius
	4	-	Index
	5	-	Tool data programmed in TOOL DEF 1 = yes, 0 = no
Active tool compensation, 200	1	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Active radius
	2	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Active length
	3	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Rounding radius R2
Active transformations, 210	1	-	Basic rotation in MANUAL OPERATION mode
	2	-	Programmed rotation with Cycle 10
	3	-	Active mirrored axis
			0: mirroring not active
			+1: X axis mirrored
			+2: Y axis mirrored
			+4: Z axis mirrored

Group name, ID no.	Number	Index	Meaning
			+64: U axis mirrored
			+128: V axis mirrored
			+256: W axis mirrored
			Combinations = sum of individual axes
	4	1	Active scaling factor in X axis
	4	2	Active scaling factor in Y axis
	4	3	Active scaling factor in Z axis
	4	7	Active scaling factor in U axis
	4	8	Active scaling factor in V axis
	4	9	Active scaling factor in W axis
	5	1	3-D ROT A axis
	5	2	3-D ROT B axis
	5	3	3-D ROT C axis
	6	-	Tilted working plane active / inactive (–1/0) in a Program Run operating mode
	7	-	Tilted working plane active / inactive (–1/0) in a Manual operating mode
Active datum shift, 220	2	1	X axis
		2	Y axis
		3	Z axis
		4	A axis
		5	B axis
		6	C axis
		7	U axis
		8	V axis
		9	W axis
Traverse range, 230	2	1 to 9	Negative software limit switch in axes 1 to 9
	3	1 to 9	Positive software limit switch in axes 1 to 9
	5	-	Software limit switch on or off: 0 = on, 1 = off

Group name, ID no.	Number	Index	Meaning
Nominal position in the REF system, 240	1	1	X axis
		2	Y axis
		3	Z axis
		4	A axis
		5	B axis
		6	C axis
		7	U axis
		8	V axis
		9	W axis
Current position in the active coordinate system, 270	1	1	X axis
		2	Y axis
		3	Z axis
		4	A axis
		5	B axis
		6	C axis
		7	U axis
		8	V axis
		9	W axis
TS triggering touch probe, 350	50	1	Touch probe type
		2	Line in the touch-probe table
	51	-	Effective length
	52	1	Effective ball radius
		2	Rounding radius
	53	1	Center offset (reference axis)
		2	Center offset (minor axis)
	54	-	Spindle-orientation angle in degrees (center offset)
	55	1	Rapid traverse
		2	Measuring feed rate

Group name, ID no.	Number	Index	Meaning
	56	1	Maximum measuring range
		2	Setup clearance
	57	1	Line in the touch-probe table
TT tool touch probe	70	1	Touch probe type
		2	Line in the touch-probe table
	71	1	Center point in reference axis (REF system)
		2	Center point in minor axis (REF system)
		3	Center point in tool axis (REF system)
	72	-	Probe contact radius
	75	1	Rapid traverse
		2	Measuring feed rate for stationary spindle
		3	Measuring feed rate for rotating spindle
	76	1	Maximum measuring range
		2	Safety clearance for linear measurement
		3	Safety clearance for radial measurement
	77	-	Spindle speed
	78	-	Probing direction
Reference point from touch probe cycle, 360	1	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Last reference point of a manual touch probe cycle, or last touch point from Cycle 0 without probe length compensation but with probe radius compensation (workpiece coordinate system)
	2	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Last reference point of a manual touch probe cycle, or last touch point from Cycle 0 without probe length or probe radius compensation (machine coordinate system)
	3	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Result of measurement of the touch probe cycles 0 and 1 without probe radius or probe length compensation
	4	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Last reference point of a manual touch probe cycle, or last touch point from Cycle 0 without probe length or stylus probe compensation (workpiece coordinate system)
	10	-	Oriented spindle stop
Value from the active datum table in the active coordinate system, 500	Line	Column	Read values

Group name, ID no.	Number	Index	Meaning
Basic transformation, 507	Line	1 to 6 (X, Y, Z, SPA, SPB, SPC)	Read the basic transformation of a preset
Axis offset, 508	Line	1 to 9 (X_OFFS, Y_OFFS, Z_OFFS, A_OFFS, B_OFFS, C_OFFS, U_OFFS, V_OFFS, W_OFFS)	Read the axis offset of a preset
Active preset, 530	1	-	Read the number of the active preset
Read data of the current tool, 950	1	-	Tool length L
	2	-	Tool radius R
	3	-	Tool radius R2
	4	-	Tool length oversize DL
	5	-	Tool radius oversize DR
	6	-	Tool radius oversize DR2
	7	-	Tool locked TL 0 = not locked, 1 = locked
	8	-	Number of the replacement tool RT
	9	-	Maximum tool age TIME1
	10	-	Maximum tool age TIME2
	11	-	Current tool age CUR. TIME
	12	-	PLC status
	13	-	Maximum tooth length LCUTS
	14	-	Maximum plunge angle ANGLE
	15	-	TT: Number of teeth CUT
	16	-	TT: Wear tolerance in length LTOL
	17	-	TT: Wear tolerance in radius RTOL
	18	-	TT: Direction of rotation DIRECT 0 = positive, –1 = negative
	19	-	TT: Offset in plane R-OFFS
	20	-	TT: Offset in length L-OFFS
	21	-	TT: Break tolerance for length LBREAK
	22	-	TT: Break tolerance for radius RBREAK

Group name, ID no.	Number	Index	Meaning
	23	-	PLC value
	24	-	Tool type TYPE 0 = milling cutter, 21 = touch probe
	27	-	Corresponding row in the touch-probe table
	32	-	Point angle
	34	-	Lift off
Touch probe cycles, 990	1	-	Approach behavior: 0 = standard behavior 1 = effective radius, set-up clearance is zero
	2	-	0 = probe monitoring off 1 = probe monitoring on
	4	-	0 = stylus not deflected 1 = stylus deflected
Execution status, 992	10	-	Block scan active 1 = yes, 0 = no
	11	-	Search phase
	14	-	Number of the last FN14 error
	16	-	Real execution active 1 = execution , 2 = simulation

Example: Assign the value of the active scaling factor for the Z axis to Q25

```
N55 D18: SYSREAD Q25 = ID210 NR4 IDX3
```

D19 PLC: Transfer values to the PLC

The **D19** function transfers up to two numerical values or Q parameters to the PLC.

Increments and units: 0.1 µm or 0.0001°

Example: Transfer the numerical value 10 (which means 1 µm or 0.001°) to the PLC

```
N56 D19 P01 +10 P02 +Q3 *
```

D20 WAIT FOR: NC and PLC synchronization

This function may only be used with the permission of your machine tool builder.

With the **D20** function you can synchronize the NC and PLC during a program run. The NC stops machining until the condition that you have programmed in the D20 block is fulfilled. The TNC can check the following PLC operands:

PLC operand	Abbreviation	Address range
Marker	**M**	0 to 4999
Input	**I**	0 to 31, 128 to 152 64 to 126 (first PL 401 B) 192 to 254 (second PL 401 B)
Output	**O**	0 to 30 32 to 62 (first PL 401 B) 64 to 94 (second PL 401 B)
Counter	**C**	48 to 79
Timer	**T**	0 to 95
Byte	**B**	0 to 4095
Word	**W**	0 to 2047
Double word	**D**	2048 to 4095

The TNC 320 uses an extended interface for communication between the PLC and NC. This is a new, symbolic Application Programmer Interface (**API**). The familiar previous PLC-NC interface is also available and can be used if desired. The machine tool builder decides whether the new or old TNC API is used. Enter the name of the symbolic operand as string to wait for the defined condition of the symbolic operand.

The following conditions are permitted in the D20 block:

Condition	Abbreviation
Equals	==

Condition	Abbreviation
Less than	**<**
Greater than	**>**
Less than or equal	**<=**
Greater than or equal	**>=**

In addition, the **D20** function is available. **WAIT FOR SYNC** is used whenever you read, for example, system data via **D18** that require synchronization with real time. The TNC stops the look-ahead calculation and executes the subsequent NC block only when the NC program has actually reached that block.

Example: Stop program run until the PLC sets marker 4095 to 1

```
N32 D20: WAIT FOR M4095==1
```

Example: Stop program run until the PLC sets the symbolic operand to 1

```
N32 D20: APISPIN[0].NN_SPICONTROLINPOS==1
```

Example: Pause internal look-ahead calculation, read current position in the X axis

```
N32 D20: WAIT FOR SYNC
N33 D18: SYSREAD Q1 = ID270 NR1 IDX1
```

D29: Transfer values to the PLC

The D29 function transfers up to eight numerical values or Q parameters to the PLC.

Increments and units: 0.1 µm or 0.0001°

Example: Transfer the numerical value 10 (which means 1 µm or 0.001°) to the PLC

```
N56 D29 P01 +10 P02 +Q3
```

D37 EXPORT

You need the D37 function if you want to create your own cycles and integrate them in the TNC. The Q parameters 0 to 99 are effective only locally. This means that the Q parameters are effective only in the program in which they were defined. With the D37 function you can export locally effective Q parameters into another (calling) program.

Example: The local Q parameter Q25 is exported

```
N56 D37 Q25
```

Example: The local Q parameters Q25 to Q30 are exported

```
N56 D37 Q25 - Q30
```

The TNC exports the value that the parameter has at the time of the EXPORT command.

The parameter is exported only to the presently calling program.

8.8 Accessing Tables with SQL Commands

Introduction

Accessing of tables is programmed on the TNC with SQL commands in **transactions**. A transaction consists of multiple SQL commands that guarantee an orderly execution of the table entries.

> Tables are configured by the machine manufacturer. Names and designations required as parameters for SQL commands are also specified.

The following **terms** are used:

- **Table:** A table consists of x columns and y rows. It is saved as a file in the File Manager of the TNC, and is addressed with the path and file name (=table name). Synonyms can also be used for addressing, as an alternative to the path and file name.
- **Columns:** The number and names of the columns are specified when configuring the table. In some SQL commands the column name is used for addressing.
- **Rows:** The number of rows is variable. You can insert new rows. There are no row numbers or other designators. However, you can select rows based on the contents of a column. Rows can only be deleted in the table editor, not by an NC program.
- **Cell:** The part of a column in a row.
- **Table entry:** Content of a cell.
- **Result set:** During a transaction, the selected columns and rows are managed in the result set. You can view the result set as a sort of "intermediate memory," which temporarily assumes the set of selected columns and rows.
- **Synonym:** This term defines a name used for a table instead of its path and file name. Synonyms are specified by the machine manufacturer in the configuration data.

A Transaction

In principle, a transaction consists of the following actions:

- Address the table (file), select rows and transfer them to the result set.
- Read rows from the result set, change rows or insert new rows.
- Conclude transaction: If changes/insertions were made, the rows from the result set are placed in the table (file).

Other actions are also necessary so that table entries can be edited in an NC program and to ensure that other changes are not made to copies of the same table rows at the same time. This results in the following **transaction sequence:**

1 A Q parameter is specified for each column to be edited. The Q parameter is assigned to a column—it is "bound" (**SQL BIND...**).

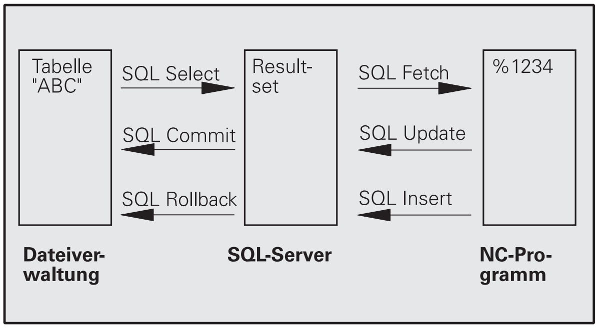
2 Address the table (file), select rows and transfer them to the result set. In addition, you define which columns are transferred to the result set (**SQL SELECT...**).

You can "lock" the selected rows. Other processes can then read these rows, but cannot change the table entries. You should always lock the selected rows when you are going to make changes (**SQL SELECT ... FOR UPDATE**).

3 Read rows from the result set, change rows or insert new rows:
– Transfer one row of the result set into the Q parameters of your NC program (**SQL FETCH...**).
– Prepare changes in the Q parameters and transfer one row from the result set (**SQL UPDATE...**).
– Prepare new table row in the Q parameters and transfer into the result set as a new row (**SQL INSERT...**).

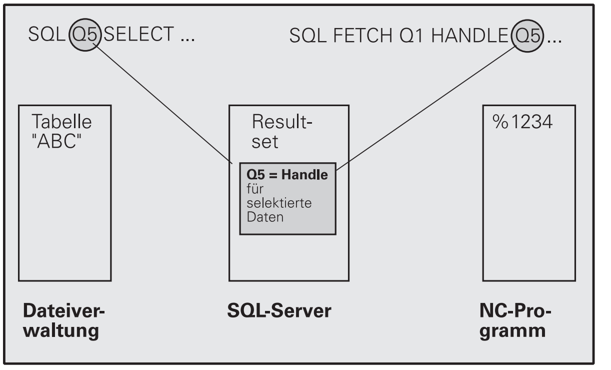
4 Conclude transaction:
– If changes/insertions were made, the data from the result set is placed in the table (file). The data is now saved in the file. Any locks are canceled, and the result set is released (**SQL COMMIT...**).
– If table entries were **not** changed or inserted (only read access), any locks are canceled and the result set is released (**SQL ROLLBACK... WITHOUT INDEX**).

Multiple transactions can be edited at the same time.



> You must conclude a transaction, even if it consists solely of read accesses. Only this guarantees that changes/insertions are not lost, that locks are canceled, and that result sets are released.

Result set



The selected rows are numbered in ascending order within the result set, starting from 0. This numbering is referred to as the **index.** The index is used for read- and write-accesses, enabling a row of the result set to be specifically addressed.

It can often be advantageous to sort the rows in the result set. Do this by specifying the table column containing the sorting criteria. Also select ascending or descending order (**SQL SELECT ... ORDER BY ...**).

The selected rows that were transferred to the result set are addressed with the **HANDLE.** All following SQL commands use the handle to refer to this "set of selected columns and rows."

When concluding a transaction, the handle is released (**SQL COMMIT...** or **SQL ROLLBACK...**). It is then no longer valid.

You can edit more than one result set at the same time. The SQL server assigns a new handle for each "Select" command.

"Binding" Q parameters to columns

The NC program does not have direct access to the table entries in the result set. The data must be transferred in Q parameters. In the other direction, the data is first prepared in the Q parameters and then transferred to the result set.

Specify with **SQL BIND ...** which table columns are mapped to which Q parameters. The Q parameters are "bound" (assigned) to the columns. Columns that are not bound to Q parameters are not included in the read-/write-processes.

If a new table row is generated with **SQL INSERT...,** the columns not bound to Q parameters are filled with default values.

Programming SQL commands

Program SQL commands in the Programming mode:

SQL

- Call the SQL functions by pressing the SQL soft key.
- Select an SQL command via soft key (see overview) or press the **SQL EXECUTE** soft key and program the SQL command.

Overview of the soft keys

Function	Soft key
SQL EXECUTE Program a "Select" command.	SQL EXECUTE
SQL BIND "Bind" a Q parameter to a table column.	SQL BIND
SQL FETCH Read table rows from the result set and save them in Q parameters.	SQL FETCH
SQL UPDATE Save data from the Q parameters in an existing table row in the result set.	SQL UPDATE
SQL INSERT Save data from the Q parameters in a new table row in the result set.	SQL INSERT
SQL COMMIT Transfer table rows from the result set into the table and conclude the transaction.	SQL COMMIT
SQL ROLLBACK ■ If **INDEX** is not programmed: Discard any changes/insertions and conclude the transaction. ■ If **INDEX** is programmed: The indexed row remains in the result set. All other rows are deleted from the result set. The transaction is **not** concluded.	SQL ROLLBACK

SQL BIND

SQL BIND binds a Q parameter to a table column. The SQL commands "Fetch," "Update" and "Insert" evaluate this binding (assignment) during data transfer between the result set and the NC program.

An **SQL BIND** command without a table or column name cancels the binding. Binding remains effective at most until the end of the NC program or subprogram.

- You can program any number of bindings. Read and write processes only take into account the columns that were entered in the "Select" command.
- **SQL BIND...** must be programmed **before** "Fetch," "Update" or "Insert" commands are programmed. You can program a "Select" command without a preceding "Bind" command.
- If in the “Select” command you include columns for which no binding is programmed, an error occurs during read/write processes (program interrupt).

SQL BIND

- **Parameter no. for result:** Q parameter that is bound (assigned) to the table column
- **Database: Column name:** Enter the table name and column name separated by a **.** (period)
  Table name: Synonym or path and file name of this table. The synonym is entered directly, whereas the path and file name are entered in single quotation marks
  Column designation: Designation of the table column as given in the configuration data

Example: Bind a Q parameter to a table column

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"
```

Example: Cancel binding

```
91 SQL BIND Q881
92 SQL BIND Q882
93 SQL BIND Q883
94 SQL BIND Q884
```

SQL SELECT

SQL SELECT selects table rows and transfers them to the result set.

The SQL server places the data in the result set row-by-row. The rows are numbered in ascending order, starting from 0. This row number, called the **INDEX,** is used in the SQL commands "Fetch" and "Update."

Enter the selection criteria in the **SQL SELECT...WHERE...** function. This lets you restrict the number of rows to be transferred. If you do not use this option, all rows in the table are loaded.

Enter the sorting criteria in the **SQL SELECT...ORDER BY...** function. Enter the column designation and the keyword for ascending/descending order. If you do not use this option, the rows are placed in random order.

Lock out the selected rows for other applications with the **SQL SELECT...FOR UPDATE** function. Other applications can continue to read these rows, but cannot change them. We strongly recommend using this option if you are making changes to the table entries.

Empty result set: If no rows match the selection criteria, the SQL server returns a valid handle but no table entries.

SQL EXECUTE

- **Parameter no. for result:** Q parameter for the handle. The SQL server returns the handle for the group of columns and rows selected with the current select command.
In case of an error (selection could not be carried out), the SQL server returns the code 1.
Code 0 identifies an invalid handle.
- **Data bank: SQL command text:** with the following elements:
  - **SELECT** (keyword):
  Name of the SQL command. Names of the table columns to be transferred. Separate column names with a **,** (comma) (see examples). Q parameters must be bound to all columns entered here.
  - **FROM** table name:
  Synonym or path and file name of this table. The synonym is entered directly, whereas the path and table name are entered in single quotation marks (see examples).
  - Optional:
  WHERE selection criteria:
  A selection criterion consists of a column name, condition (see table) and comparator. Link selection criteria with logical AND or OR. Program the comparator directly or with a Q parameter. A Q parameter is introduced with a colon and placed in single quotation marks (see example).
  - Optional:
  ORDER BY column name **ASC** to sort in ascending order—or
  ORDER BY column name **DESC** to sort in descending order.
  If neither ASC nor DESC are programmed, then ascending order is used as the default setting. The TNC places the selected rows in the indicated column.
  - Optional:
  FOR UPDATE (keyword):
  The selected rows are locked against write-accesses from other processes.

Example: Select all table rows

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE"
```

Example: Selection of table rows with the WHERE function

```
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE WHERE MEAS_NO<20"
```

Example: Selection of table rows with the WHERE function and Q parameters

```
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE WHERE
MEAS_NO==:'Q11'"
```

Example: Table name defined with path and file name

```
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM 'V:\TABLE\TAB_EXAMPLE' WHERE
MEAS_NO<20"
```

Condition	Programming
Equal to	= ==
Not equal to	!= <>
Less than	<
Less than or equal to	<=
Greater than	>
Greater than or equal to	>=
Linking multiple conditions:	
Logical AND	AND
Logical OR	OR

SQL FETCH

SQL FETCH reads the row addressed with **INDEX** from the result set, and places the table entries in the bound (assigned) Q parameters. The result set is addressed with the **HANDLE.**

SQL FETCH takes into account all columns entered in the "Select" command.

SQL FETCH

- **Parameter no. for result:** Q parameter in which the SQL server reports the result:
  0: No error occurred.
  1: Error occurred (incorrect handle or index too large)
- **Data bank: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**)
- **Data bank: Index for SQL result:** Row number within the result set. The table entries of this row are read and are transferred into the bound Q parameters. If you do not enter an index, the first row is read (n=0).
  Either enter the row number directly or program the Q parameter containing the index

Example: Row number is transferred in a Q parameter

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
```

Example: Row number is programmed directly

```
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX5
```

SQL UPDATE

SQL UPDATE transfers the data prepared in the Q parameters into the row of the result set addressed with **INDEX.** The existing row in the result set is completely overwritten.

SQL UPDATE takes into account all columns entered in the "Select" command.

SQL UPDATE

- **Parameter no. for result:** Q parameter in which the SQL server reports the result:
  0: No error occurred.
  1: Error occurred (incorrect handle, index too large, value outside of value range or incorrect data format)
- **Data bank: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**)
- **Data bank: Index for SQL result:** Row number within the result set. The table entries prepared in the Q parameters are written to this row. If you do not enter an index, the first row is written to (n=0). Either enter the row number directly or program the Q parameter containing the index

Example: Row number is transferred in a Q parameter

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
. . .
40 SQL UPDATE Q1 HANDLE Q5 INDEX+Q2
```

Example: Row number is programmed directly

```
. . .
40 SQL UPDATE Q1 HANDLE Q5 INDEX5
```

SQL INSERT

SQL INSERT generates a new row in the result set and transfers the data prepared in the Q parameters into the new row.

SQL INSERT takes into account all columns entered in the "Select" command. Table columns not entered in the "Select" command are filled with default values.

SQL INSERT

- **Parameter no. for result:** Q parameter in which the SQL server reports the result:
  0: No error occurred.
  1: Error occurred (incorrect handle, value outside of value range or incorrect data format)
- **Data bank: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**)

Example: Row number is transferred in a Q parameter

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE"
. . .
40 SQL INSERT Q1 HANDLE Q5
```

SQL COMMIT

SQL COMMIT transfers all rows in the result set back to the table. A lock set with **SELECT...FOR UPDATE** is canceled.

The handle given in the **SQL SELECT** command loses its validity.

SQL COMMIT

- **Parameter no. for result:** Q parameter in which the SQL server reports the result:
  0: No error occurred.
  1: Error occurred (incorrect handle or equal entries in columns requiring unique entries)
- **Data bank: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**)

Example:

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
. . .
40 SQL UPDATE Q1 HANDLE Q5 INDEX+Q2
. . .
50 SQL COMMIT Q1 HANDLE Q5
```

SQL ROLLBACK

How **SQL ROLLBACK** is executed depends on whether **INDEX** is programmed:

- If **INDEX** is not programmed: The result set is **not** written back to the table (any changes/insertions are discarded). The transaction is closed and the handle given in the **SQL SELECT** command loses its validity. Typical application: Ending a transaction solely containing read-accesses.
- If **INDEX** is programmed: The indexed row remains. All other rows are deleted from the result set. The transaction is **not** concluded. A lock set with **SELECT...FOR UPDATE** remains for the indexed row. For all other rows it is reset.

SQL ROLLBACK

- **Parameter no. for result:** Q parameter in which the SQL server reports the result:
  0: No error occurred.
  1: Error occurred (incorrect handle)
- **Data bank: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**)
- **Data bank: Index for SQL result:** Row that is to remain in the result set. Either enter the row number directly or program the Q parameter containing the index

Example:

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
. . .
50 SQL ROLLBACK Q1 HANDLE Q5
```

8.9 Entering Formulas Directly

Entering formulas

You can enter mathematical formulas that include several operations directly into the part program by soft key.

Press the FORMULA soft key to call the mathematical functions. The TNC displays the following soft keys in several soft-key rows:

Mathematical function	Soft key
Addition Example: **Q10 = Q1 + Q5**	+
Subtraction Example: **Q25 = Q7 – Q108**	-
Multiplication Example: **Q12 = 5 * Q5**	*
Division Example: **Q25 = Q1 / Q2**	/
Opening parenthesis Example: **Q12 = Q1 * (Q2 + Q3)**	(
Closing parenthesis Example: **Q12 = Q1 * (Q2 + Q3)**	)
Square of a value Example: **Q15 = SQ 5**	SQ
Square root Example: **Q22 = SQRT 25**	SQRT
Sine of an angle Example: **Q44 = SIN 45**	SIN
Cosine of an angle Example: **Q45 = COS 45**	COS
Tangent of an angle Example: **Q46 = TAN 45**	TAN
Arc sine Inverse of the sine. Determines the angle from the ratio of the side opposite the angle and the hypotenuse. Example: **Q10 = ASIN 0.75**	ASIN
Arc cosine Inverse of the cosine. Determines the angle from the ratio of the side adjacent to the angle and the hypotenuse. Example: **Q11 = ACOS Q40**	ACOS

Mathematical function	Soft key
Arc tangent Inverse of the tangent. Determines the angle from the ratio of the opposite side to the adjacent side. Example: **Q12 = ATAN Q50**	ATAN
Powers of values Example: **Q15 = 3^3**	^
Constant "pi" (3.14159) Example: **Q15 = PI**	PI
Natural logarithm (LN) of a number Base 2.7183 Example: **Q15 = LN Q11**	LN
Logarithm of a number, base 10 Example: **Q33 = LOG Q22**	LOG
Exponential function, 2.7183 to the power of *n* Example: **Q1 = EXP Q12**	EXP
Negate (multiplication by –1) Example: **Q2 = NEG Q1**	NEG
Truncate decimal places Form an integer Example: **Q3 = INT Q42**	INT
Absolute value of a number Example: **Q4 = ABS Q22**	ABS
Truncate places before the decimal point Form a fraction Example: **Q5 = FRAC Q23**	FRAC
Check algebraic sign of a number Example: **Q12 = SGN Q50** If result for Q12 = 1, then Q50 >= 0 If result for Q12 = –1, then Q50 < 0	SGN
Calculate modulo value (division remainder) Example: **Q12 = 400 % 360** Result: Q12 = 40	%

Rules for formulas

Mathematical formulas are programmed according to the following rules:

Higher-level operations are performed first

```
12  Q1 = 5 * 3 + 2 * 10 = 35
```

1st calculation: 5 * 3 = 15
2nd calculation: 2 * 10 = 20
3rd calculation: 15 + 20 = 35

or

```
13  Q2 = SQ 10 - 3^3 = 73
```

1st calculation: 10 squared = 100
2nd calculation: 3 to the power of 3 = 27
3rd calculation: 100 – 27 = 73

Distributive law

Law for calculating with parentheses

a * (b + c) = a * b + a * c

Programming example

Calculate an angle with the arc tangent from the opposite side (Q12) and adjacent side (Q13); then store in Q25.

Q FORMULA — To select the formula entering function, press the Q key and the FORMULA soft key, or use the shortcut:

Q — Press the Q key on the ASCII keyboard.

PARAMETER NO. FOR RESULT?

ENT 25 — Enter the parameter number

▷ ATAN — Shift the soft-key row and select the arc tangent function

◁ (— Shift the soft-key row and open the parentheses

Q 12 — Enter Q parameter number 12

/ — Select division

Q 13 — Enter Q parameter number 13

) END — Close parentheses and conclude formula entry

Example NC block

```
37  Q25 = ATAN (Q12/Q13)
```

8.10 String Parameters

String processing functions

You can use the **QS** parameters to create variable character strings.

You can assign a linear sequence of characters (letters, numbers, special characters and spaces) up to a length of 256 characters to a string parameter. You can also check and process the assigned or imported values by using the functions described below. As in Q-parameter programming, you can use a total of 2000 QS parameters (see also "Principle and Overview" on page 202).

The STRING FORMULA and FORMULA Q parameter functions contain various functions for processing the string parameters.

STRING FORMULA functions	Soft key	Page
Assigning string parameters	STRING	Page 246
Chain-linking string parameters		Page 246
Converting a numerical value to a string parameter	TOCHAR	Page 248
Copying a substring from a string parameter	SUBSTR	Page 249

FORMULA string functions	Soft key	Page
Converting a string parameter to a numerical value	TONUMB	Page 250
Checking a string parameter	INSTR	Page 251
Finding the length of a string parameter	STRLEN	Page 252
Comparing alphabetic priority	STRCOMP	Page 253

When you use a STRING FORMULA, the result of the arithmetic operation is always a string. When you use the FORMULA function, the result of the arithmetic operation is always a numeric value.

Assigning string parameters

You have to assign a string variable before you use it. Use the **DECLARE STRING** command to do so.

SPEC FCT

- Show the soft-key row with special functions

PROGRAM FUNCTIONS

- Select the menu for defining various plain-language functions

STRING FUNCTIONS

- Select string functions

DECLARE STRING

- Select the **DECLARE STRING** function

Example NC block:

```
N37 DECLARE STRING QS10 = "WORKPIECE"
```

Chain-linking string parameters

With the concatenation operator (string parameter || string parameter) you can make a chain of two or more string parameters.

SPEC FCT

- Show the soft-key row with special functions

PROGRAM FUNCTIONS

- Select the menu for defining various plain-language functions

STRING FUNCTIONS

- Select string functions

STRING FORMULA

- Select the STRING FORMULA function
- Enter the number of the string parameter in which the TNC is to save the concatenated string. Confirm with the ENT key
- Enter the number of the string parameter in which the **first** substring is saved. Confirm with the ENT key: The TNC displays the concatenation symbol ||
- Confirm your entry with the ENT key
- Enter the number of the string parameter in which the **second** substring is saved. Confirm with the ENT key
- Repeat the process until you have selected all the required substrings. Conclude with the END key

Example: QS10 is to include the complete text of QS12, QS13 and QS14

```
N37 QS10 =  QS12 || QS13 || QS14
```

Parameter contents:

- **QS12: Workpiece**
- **QS13: Status:**
- **QS14: Scrap**
- **QS10: Workpiece Status: Scrap**

Converting a numerical value to a string parameter

With the **TOCHAR** function, the TNC converts a numerical value to a string parameter. This enables you to chain numerical values with string variables.

SPEC FCT

- Show the soft-key row with special functions

PROGRAM FUNCTIONS

- Select the menu for defining various plain-language functions

STRING FUNCTIONS

- Select string functions

STRING FORMULA

- Select the STRING FORMULA function
- Enter the number of the string parameter in which the TNC is to save the converted value. Confirm with the ENT key

TOCHAR

- Select the function for converting a numerical value to a string parameter
- Enter the number or the desired Q parameter to be converted, and confirm with the ENT key
- If desired, enter the number of decimal places that the TNC should convert, and confirm with the ENT key
- Close the parenthetical expression with the ENT key and confirm your entry with the END key

Example: Convert parameter Q50 to string parameter QS11, use 3 decimal places

```
N37 QS11 = TOCHAR ( DAT+Q50 DECIMALS3 )
```

Copying a substring from a string parameter

With the **SUBSTR** function you can copy a definable range from a string parameter.

Q

- Select Q parameter functions

STRING FORMULA

- Select the STRING FORMULA function
- Enter the number of the string parameter in which the TNC is to save the copied string. Confirm with the ENT key

SUBSTR

- Select the function for copying a substring
- Enter the number of the QS parameter from which the substring is to be copied. Confirm with the ENT key
- Enter the number of the place starting from which to copy the substring, and confirm with the ENT key
- Enter the number of characters to be copied, and confirm with the ENT key
- Close the parenthetical expression with the ENT key and confirm your entry with the END key

Remember that the first character of a text sequence starts internally with the zeroth place.

Example: A four-character substring (LEN4) is read from the string parameter QS10 beginning with the third character (BEG2)

```
N37 QS13 = SUBSTR ( SRC_QS10 BEG2 LEN4 )
```

Converting a string parameter to a numerical value

The **TONUMB** function converts a string parameter to a numerical value. The value to be converted should be only numerical.

The QS parameter must contain only a numerical value. Otherwise the TNC will output an error message.

Q

- Select Q parameter functions

FORMULA

- Select the FORMULA function
- Enter the number of the parameter in which the TNC is to save the numerical value. Confirm with the ENT key

◁

- Shift the soft-key row

TONUMB

- Select the function for converting a string parameter to a numerical value
- Enter the number of the Q parameter to be converted, and confirm with the ENT key
- Close the parenthetical expression with the ENT key and confirm your entry with the END key

Example: Convert string parameter QS11 to a numerical parameter Q82

```
N37 Q82 = TONUMB ( SRC_QS11 )
```

Checking a string parameter

With the **INSTR** function you can check whether a string parameter is contained in another string parameter.

Q

- Select Q parameter functions

FORMULA

- Select the FORMULA function
- Enter the number of the Q parameter in which the TNC is to save the place at which the search text begins. Confirm with the ENT key

◁

- Shift the soft-key row

INSTR

- Select the function for checking a string parameter
- Enter the number of the QS parameter in which the text to be searched for is saved. Confirm with the ENT key
- Enter the number of the QS parameter to be searched, and confirm with the ENT key
- Enter the number of the place starting from which the TNC is to search the substring, and confirm with the ENT key
- Close the parenthetical expression with the ENT key and confirm your entry with the END key

Remember that the first character of a text sequence starts internally with the zeroth place.

If the TNC cannot find the required substring, it will save the total length of the string to be searched (counting starts at 1) in the result parameter.

If the substring is found in more than one place, the TNC returns the first place at which it finds the substring.

Example: Search through QS13 for the text saved in parameter QS10. Begin the search at the third place.

```
N37 Q50 = INSTR ( SRC_QS10 SEA_QS13 BEG2 )
```

Finding the length of a string parameter

The **STRLEN** function returns the length of the text saved in a selectable string parameter.

Q

- Select Q parameter functions

FORMULA

- Select the FORMULA function
- Enter the number of the Q parameter in which the TNC is to save the ascertained string length. Confirm with the ENT key

◁

- Shift the soft-key row

STRLEN

- Select the function for finding the text length of a string parameter
- Enter the number of the QS parameter whose length the TNC is to ascertain, and confirm with the ENT key
- Close the parenthetical expression with the ENT key and confirm your entry with the END key

Example: Find the length of QS15

```
N37 Q52 = STRLEN ( SRC_QS15 )
```

Comparing alphabetic priority

With the **STRCOMP** function you can compare string parameters for alphabetic priority.

- Select Q parameter functions
- Select the FORMULA function
- Enter the number of the Q parameter in which the TNC is to save the result of comparison. Confirm with the ENT key
- Shift the soft-key row
- Select the function for comparing string parameters
- Enter the number of the first QS parameter to be compared, and confirm with the ENT key
- Enter the number of the second QS parameter to be compared, and confirm with the ENT key
- Close the parenthetical expression with the ENT key and confirm your entry with the END key

The TNC returns the following results:

- **0**: The compared QS parameters are identical.
- **+1**: The first QS parameter **precedes** the second QS parameter alphabetically.
- **–1**: The first QS parameter **follows** the second QS parameter alphabetically.

Example: QS12 and QS14 are compared for alphabetic priority

```
N37 Q52 = STRCOMP ( SRC_QS12 SEA_QS14 )
```

Reading machine parameters

Use the **CFGREAD** function to read out TNC machine parameters as numerical values or as strings.

In order to read out a machine parameter, you must use the TNC's configuration editor to determine the parameter name, parameter object, and, if they have been assigned, the group name and index:

Type	Meaning	Example	Symbol
Key	Group name of the machine parameter (if assigned)	CH_NC	
Entity	Parameter object (the name starts with "**Cfg**...")	CfgGeoCycle	
Attribute	Name of the machine parameter	displaySpindleErr	
Index	List index of a machine parameter (if assigned)	[0]	

If you are in the configuration editor for the user parameters, you can change the display of the existing parameters. In the default setting, the parameters are displayed with short, explanatory texts. To display the actual system names of the parameters, press the key for the screen layout and then the SHOW SYSTEM NAME soft key. Follow the same procedure to return to the standard display.

Each time you want to interrogate a machine parameter with the **CFGREAD** function, you must first define a QS parameter with attribute, entity and key.

The following parameters are read in the **CFGREAD** function's dialog:

- **KEY_QS**: Group name (key) of the machine parameter
- **TAG_QS**: Object name (entity) of the machine parameter
- **ATR_QS**: Name (attribute) of the machine parameter
- **IDX**: Index of the machine parameter

Reading a string of a machine parameter

In order to store the content of a machine parameter as a string in a QS parameter:

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Select the menu for defining various plain-language functions

STRING FUNCTIONS
- Select string functions

STRING FORMULA
- Select the STRING FORMULA function
- Enter the number of the string parameter in which the TNC is to save the machine parameter. Confirm with the ENT key
- Select the CFGREAD function
- Enter the numbers of the string parameters for the key, entity and attribute, then confirm with the ENT key
- Enter the number for the index, or skip the dialog with NO ENT, whichever applies
- Close the parenthetical expression with the ENT key and confirm your entry with the END key

Example: Read as a string the axis designation of the fourth axis

Parameter settings in the configuration editor

DisplaySettings

    CfgDisplayData
        axisDisplayOrder
            [0] to [5]

`14 DECLARE STRING QS11 = ""`	Assign string parameter for key
`15 DECLARE STRING QS12 = "CfgDisplayData"`	Assign string parameter for entity
`16 DECLARE STRING QS13 = "axisDisplayOrder"`	Assign string parameter for parameter name
`17 QS1 = CFGREAD( KEY_QS11 TAG_QS12 ATR_QS13 IDX3 )`	Read out machine parameter

Reading a numerical value of a machine parameter

In order to store the value of a machine parameter as a numerical value in a Q parameter:

Q

- Select Q parameter functions

FORMULA

- Select the FORMULA function
- Enter the number of the Q parameter in which the TNC is to save the machine parameter. Confirm with the ENT key
- Select the CFGREAD function
- Enter the numbers of the string parameters for the key, entity and attribute, then confirm with the ENT key
- Enter the number for the index, or skip the dialog with NO ENT, whichever applies
- Close the parenthetical expression with the ENT key and confirm your entry with the END key

Example: Read overlap factor as Q parameter

Parameter settings in the configuration editor
ChannelSettings CH_NC CfgGeoCycle pocketOverlap

`14 DECLARE STRING QS11 = "CH_NC"`	Assign string parameter for key
`15 DECLARE STRING QS12 = "CfgGeoCycle"`	Assign string parameter for entity
`16 DECLARE STRING QS13 = "pocketOverlap"`	Assign string parameter for parameter name
`17 Q50 = CFGREAD( KEY_QS11 TAG_QS12 ATR_QS13 )`	Read out machine parameter

8.11 Preassigned Q Parameters

The Q parameters Q100 to Q199 are assigned values by the TNC. The following types of information are assigned to Q parameters:

- Values from the PLC
- Tool and spindle data
- Data on operating status
- Results of measurements from touch probe cycles etc.

The TNC saves the values for the preassigned Q parameters Q108, Q114 and Q115 to Q117 in the unit of measure used by the active program.

> Do not use preassigned Q parameters (or QS parameters) between **Q100** and **Q199 (QS100** and **QS199)** as calculation parameters in NC programs. Otherwise you might receive undesired results.

Values from the PLC: Q100 to Q107

The TNC uses the parameters Q100 to Q107 to transfer values from the PLC to an NC program.

Active tool radius: Q108

The active value of the tool radius is assigned to Q108. Q108 is calculated from:

- Tool radius R (tool table or **G99** block)
- Delta value DR from the tool table
- Delta value DR from the **T** block

> The TNC remembers the current tool radius even if the power is interrupted.

Tool axis: Q109

The value of Q109 depends on the current tool axis:

Tool axis	Parameter value
No tool axis defined	Q109 = –1
X axis	Q109 = 0
Y axis	Q109 = 1
Z axis	Q109 = 2
U axis	Q109 = 6
V axis	Q109 = 7
W axis	Q109 = 8

Spindle status: Q110

The value of the parameter Q110 depends on the M function last programmed for the spindle.

M function	Parameter value
No spindle status defined	Q110 = –1
M3: Spindle ON, clockwise	Q110 = 0
M4: Spindle ON, counterclockwise	Q110 = 1
M5 after M3	Q110 = 2
M5 after M4	Q110 = 3

Coolant on/off: Q111

M function	Parameter value
M8: Coolant ON	Q111 = 1
M9: Coolant OFF	Q111 = 0

Overlap factor: Q112

The overlap factor for pocket milling (pocketOverlap) is assigned to Q112.

Unit of measurement for dimensions in the program: Q113

During nesting with PGM CALL, the value of the parameter Q113 depends on the dimensional data of the program from which the other programs are called.

Dimensional data of the main program	Parameter value
Metric system (mm)	Q113 = 0
Inch system (inches)	Q113 = 1

Tool length: Q114

The current value for the tool length is assigned to Q114.

The TNC remembers the current tool length even if the power is interrupted.

Coordinates after probing during program run

The parameters Q115 to Q119 contain the coordinates of the spindle position at the moment of contact during programmed measurement with the 3-D touch probe. The coordinates refer to the datum point that is active in the Manual operating mode.

The length of the stylus and the radius of the ball tip are not compensated in these coordinates.

Coordinate axis	Parameter value
X axis	Q115
Y axis	Q116
Z axis	Q117
4th axis machine-dependent	Q118
5th axis machine-dependent	Q119

Deviation between actual value and nominal value during automatic tool measurement with the TT 130

Deviation of actual from nominal value	Parameter value
Tool length	Q115
Tool radius	Q116

Tilting the working plane with mathematical angles: rotary axis coordinates calculated by the TNC

Coordinates	Parameter value
A axis	Q120
B axis	Q121
C axis	Q122

Measurement results from touch probe cycles (see also User's Manual for Touch Probe Cycles)

Measured actual values	Parameter value
Angle of a straight line	Q150
Center in the major axis	Q151
Center in the minor axis	Q152
Diameter	Q153
Pocket length	Q154
Pocket width	Q155
Length of the axis selected in the cycle	Q156
Position of the centerline	Q157
Angle of the A axis	Q158
Angle of the B axis	Q159
Coordinate of the axis selected in the cycle	Q160

Measured deviation	Parameter value
Center in the major axis	Q161
Center in the minor axis	Q162
Diameter	Q163
Pocket length	Q164
Pocket width	Q165
Measured length	Q166
Position of the centerline	Q167

Determined space angle	Parameter value
Rotation about the A axis	Q170
Rotation about the B axis	Q171
Rotation about the C axis	Q172

Workpiece status	Parameter value
Good	Q180
Rework	Q181
Scrap	Q182

Measured deviation with Cycle 440	Parameter value
X axis	Q185
Y axis	Q186
Z axis	Q187
Marker for cycles	Q188

Tool measurement with the BLUM laser	Parameter value
Reserved	Q190
Reserved	Q191
Reserved	Q192
Reserved	Q193

Reserved for internal use	Parameter value
Marker for cycles	Q195
Marker for cycles	Q196
Marker for cycles (machining patterns)	Q197
Number of the last active measuring cycle	Q198

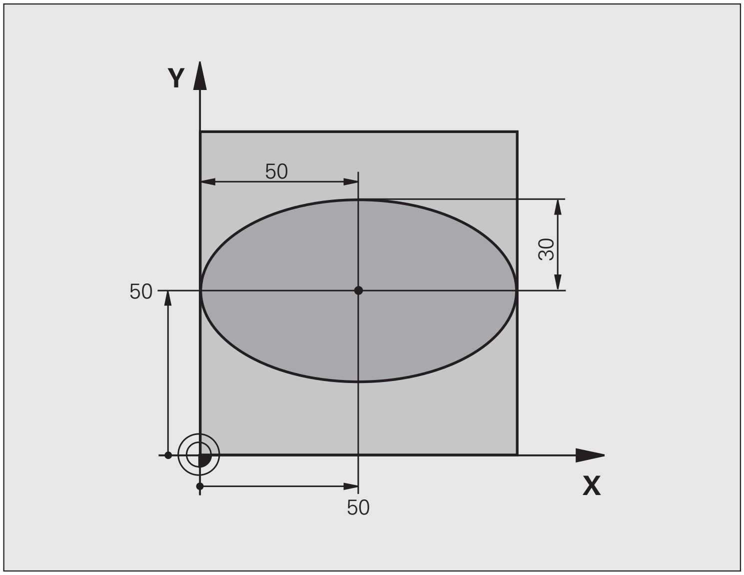
Status of tool measurement with TT	Parameter value
Tool within tolerance	Q199 = 0,0
Tool is worn (LTOL/RTOL is exceeded)	Q199 = 1,0
Tool is broken (LBREAK/RBREAK is exceeded)	Q199 = 2,0

8.12 Programming Examples

Example: Ellipse

Program sequence

- The contour of the ellipse is approximated by many short lines (defined in Q7). The more calculation steps you define for the lines, the smoother the curve becomes.
- The machining direction can be altered by changing the entries for the starting and end angles in the plane:
  Clockwise machining direction:
  starting angle > end angle
  Counterclockwise machining direction:
  starting angle < end angle
- The tool radius is not taken into account.



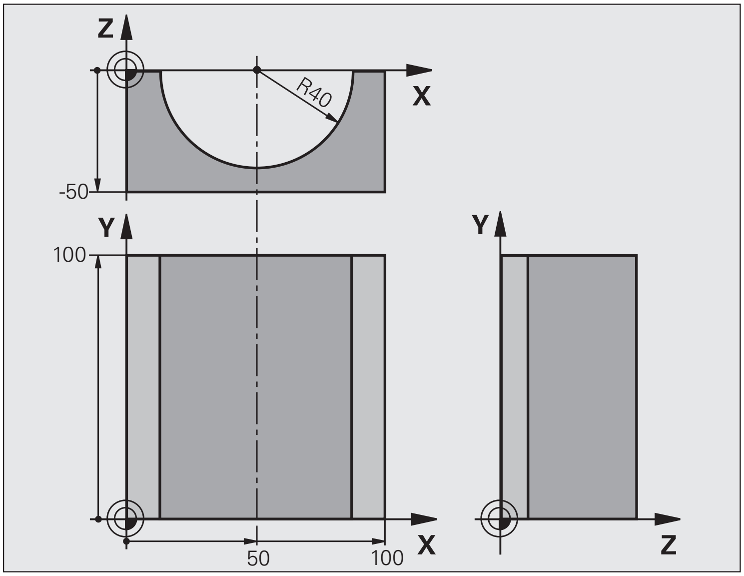
%ELLIPSE G71 *	
N10 D00 Q1 P01 +50 *	Center in X axis
N20 D00 Q2 P01 +50 *	Center in Y axis
N30 D00 Q3 P01 +50 *	Semiaxis in X
N40 D00 Q4 P01 +30 *	Semiaxis in Y
N50 D00 Q5 P01 +0 *	Starting angle in the plane
N60 D00 Q6 P01 +360 *	End angle in the plane
N70 D00 Q7 P01 +40 *	Number of calculation steps
N80 D00 Q8 P01 +30 *	Rotational position of the ellipse
N90 D00 Q9 P01 +5 *	Milling depth
N100 D00 Q10 P01 +100 *	Feed rate for plunging
N110 D00 Q11 P01 +350 *	Feed rate for milling
N120 D00 Q12 P01 +2 *	Set-up clearance for pre-positioning
N130 G30 G17 X+0 Y+0 Z-20 *	Definition of workpiece blank
N140 G31 G90 X+100 Y+100 Z+0 *	
N150 T1 G17 S4000 *	Tool call
N160 G00 G40 G90 Z+250 *	Retract the tool
N170 L10.0 *	Call machining operation

N180 G00 Z+250 M2 *	Retract in the tool axis, end program
N190 G98 L10 *	Subprogram 10: Machining operation
N200 G54 X+Q1 Y+Q2 *	Shift datum to center of ellipse
N210 G73 G90 H+Q8 *	Account for rotational position in the plane
N220 Q35 = (Q6 - Q5) / Q7 *	Calculate angle increment
N230 D00 Q36 P01 +Q5 *	Copy starting angle
N240 D00 Q37 P01 +0 *	Set counter
N250 Q21 = Q3 * COS Q36 *	Calculate X coordinate for starting point
N260 Q22 = Q4 * SIN Q36 *	Calculate Y coordinate for starting point
N270 G00 G40 X+Q21 Y+Q22 M3 *	Move to starting point in the plane
N280 Z+Q12 *	Pre-position in spindle axis to set-up clearance
N290 G01 Z-Q9 FQ10 *	Move to working depth
N300 G98 L1 *	
N310 Q36 = Q36 + Q35 *	Update the angle
N320 Q37 = Q37 + 1 *	Update the counter
N330 Q21 = Q3 * COS Q36 *	Calculate the current X coordinate
N340 Q22 = Q4 * SIN Q36 *	Calculate the current Y coordinate
N350 G01 X+Q21 Y+Q22 FQ11 *	Move to next point
N360 D12 P01 +Q37 P02 +Q7 P03 1 *	Unfinished? If not finished, return to LBL 1
N370 G73 G90 H+0 *	Reset the rotation
N380 G54 X+0 Y+0 *	Reset the datum shift
N390 G00 G40 Z+Q12 *	Move to set-up clearance
N400 G98 L0 *	End of subprogram
N99999999 %ELLIPSE G71 *	

Example: Concave cylinder machined with spherical cutter

Program sequence

- This program functions only with a spherical cutter. The tool length refers to the sphere center.
- The contour of the cylinder is approximated by many short line segments (defined in Q13). The more line segments you define, the smoother the curve becomes.
- The cylinder is milled in longitudinal cuts (here: parallel to the Y axis).
- The machining direction can be altered by changing the entries for the starting and end angles in space:
  Clockwise machining direction:
  starting angle > end angle
  Counterclockwise machining direction:
  starting angle < end angle
- The tool radius is compensated automatically.



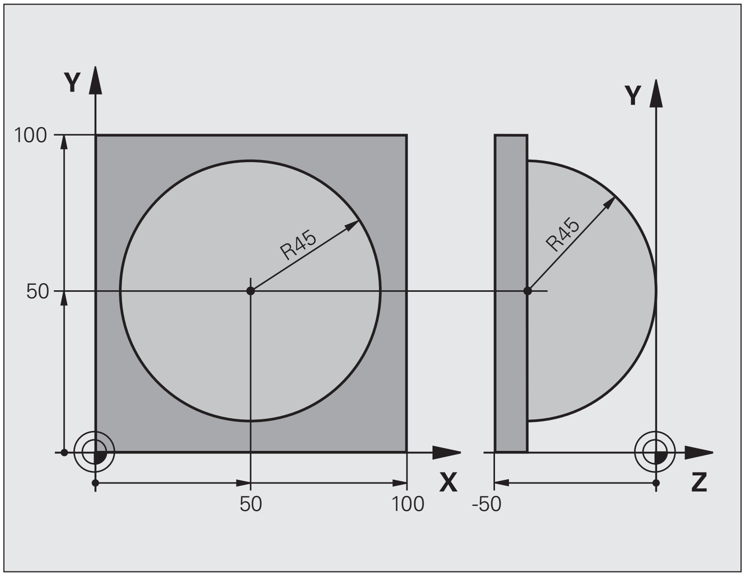
`%CYLIN G71 *`	
`N10 D00 Q1 P01 +50 *`	Center in X axis
`N20 D00 Q2 P01 +0 *`	Center in Y axis
`N30 D00 Q3 P01 +0 *`	Center in Z axis
`N40 D00 Q4 P01 +90 *`	Starting angle in space (Z/X plane)
`N50 D00 Q5 P01 +270 *`	End angle in space (Z/X plane)
`N60 D00 Q6 P01 +40 *`	Cylinder radius
`N70 D00 Q7 P01 +100 *`	Length of the cylinder
`N80 D00 Q8 P01 +0 *`	Rotational position in the X/Y plane
`N90 D00 Q10 P01 +5 *`	Allowance for cylinder radius
`N100 D00 Q11 P01 +250 *`	Feed rate for plunging
`N110 D00 Q12 P01 +400 *`	Feed rate for milling
`N120 D00 Q13 P01 +90 *`	Number of cuts
`N130 G30 G17 X+0 Y+0 Z-50 *`	Definition of workpiece blank
`N140 G31 G90 X+100 Y+100 Z+0 *`	
`N150 T1 G17 S4000 *`	Tool call
`N160 G00 G40 G90 Z+250 *`	Retract the tool
`N170 L10.0 *`	Call machining operation
`N180 D00 Q10 P01 +0 *`	Reset allowance
`N190 L10.0`	Call machining operation

N200 G00 G40 Z+250 M2 *	Retract in the tool axis, end program
N210 G98 L10 *	Subprogram 10: Machining operation
N220 Q16 = Q6 - Q10 - Q108 *	Account for allowance and tool, based on the cylinder radius
N230 D00 Q20 P01 +1 *	Set counter
N240 D00 Q24 P01 +Q4 *	Copy starting angle in space (Z/X plane)
N250 Q25 = (Q5 - Q4) / Q13 *	Calculate angle increment
N260 G54 X+Q1 Y+Q2 Z+Q3 *	Shift datum to center of cylinder (X axis)
N270 G73 G90 H+Q8 *	Account for rotational position in the plane
N280 G00 G40 X+0 Y+0 *	Pre-position in the plane to the cylinder center
N290 G01 Z+5 F1000 M3 *	Pre-position in the tool axis
N300 G98 L1 *	
N310 I+0 K+0 *	Set pole in the Z/X plane
N320 G11 R+Q16 H+Q24 FQ11 *	Move to starting position on cylinder, plunge-cutting obliquely into the material
N330 G01 G40 Y+Q7 FQ12 *	Longitudinal cut in Y+ direction
N340 D01 Q20 P01 +Q20 P02 +1 *	Update the counter
N350 D01 Q24 P01 +Q24 P02 +Q25 *	Update solid angle
N360 D11 P01 +Q20 P02 +Q13 P03 99 *	Finished? If finished, jump to end
N370 G11 R+Q16 H+Q24 FQ11 *	Move in an approximated “arc” for the next longitudinal cut
N380 G01 G40 Y+0 FQ12 *	Longitudinal cut in Y– direction
N390 D01 Q20 P01 +Q20 P02 +1 *	Update the counter
N400 D01 Q24 P01 +Q24 P02 +Q25 *	Update solid angle
N410 D12 P01 +Q20 P02 +Q13 P03 1 *	Unfinished? If not finished, return to LBL 1
N420 G98 L99 *	
N430 G73 G90 H+0 *	Reset the rotation
N440 G54 X+0 Y+0 Z+0 *	Reset the datum shift
N450 G98 L0 *	End of subprogram
N99999999 %CYLIN G71 *	

Example: Convex sphere machined with end mill

Program sequence

- This program requires an end mill.
- The contour of the sphere is approximated by many short lines (in the Z/X plane, defined in Q14). The smaller you define the angle increment, the smoother the curve becomes.
- You can determine the number of contour cuts through the angle increment in the plane (defined in Q18).
- The tool moves upward in three-dimensional cuts.
- The tool radius is compensated automatically.



%SPHERE G71 *	
N10 D00 Q1 P01 +50 *	Center in X axis
N20 D00 Q2 P01 +50 *	Center in Y axis
N30 D00 Q4 P01 +90 *	Starting angle in space (Z/X plane)
N40 D00 Q5 P01 +0 *	End angle in space (Z/X plane)
N50 D00 Q14 P01 +5 *	Angle increment in space
N60 D00 Q6 P01 +45 *	Sphere radius
N70 D00 Q8 P01 +0 *	Starting angle of rotational position in the X/Y plane
N80 D00 Q9 P01 +360 *	End angle of rotational position in the X/Y plane
N90 D00 Q18 P01 +10 *	Angle increment in the X/Y plane for roughing
N100 D00 Q10 P01 +5 *	Allowance in sphere radius for roughing
N110 D00 Q11 P01 +2 *	Set-up clearance for pre-positioning in the tool axis
N120 D00 Q12 P01 +350 *	Feed rate for milling
N130 G30 G17 X+0 Y+0 Z-50 *	Definition of workpiece blank
N140 G31 G90 X+100 Y+100 Z+0 *	
N150 T1 G17 S4000 *	Tool call
N160 G00 G40 G90 Z+250 *	Retract the tool

N170 L10.0 *	Call machining operation
N180 D00 Q10 P01 +0 *	Reset allowance
N190 D00 Q18 P01 +5 *	Angle increment in the X/Y plane for finishing
N200 L10.0 *	Call machining operation
N210 G00 G40 Z+250 M2 *	Retract in the tool axis, end program
N220 G98 L10 *	Subprogram 10: Machining operation
N230 D01 Q23 P01 +Q11 P02 +Q6 *	Calculate Z coordinate for pre-positioning
N240 D00 Q24 P01 +Q4 *	Copy starting angle in space (Z/X plane)
N250 D01 Q26 P01 +Q6 P02 +Q108 *	Compensate sphere radius for pre-positioning
N260 D00 Q28 P01 +Q8 *	Copy rotational position in the plane
N270 D01 Q16 P01 +Q6 P02 -Q10 *	Account for allowance in the sphere radius
N280 G54 X+Q1 Y+Q2 Z-Q16 *	Shift datum to center of sphere
N290 G73 G90 H+Q8 *	Account for starting angle of rotational position in the plane
N300 G98 L1 *	Pre-position in the tool axis
N310 I+0 J+0 *	Set pole in the X/Y plane for pre-positioning
N320 G11 G40 R+Q26 H+Q8 FQ12 *	Pre-position in the plane
N330 I+Q108 K+0 *	Set pole in the Z/X plane, offset by the tool radius
N340 G01 Y+0 Z+0 FQ12 *	Move to working depth
N350 G98 L2 *	
N360 G11 G40 R+Q6 H+Q24 FQ12 *	Move upward in an approximated “arc”
N370 D02 Q24 P01 +Q24 P02 +Q14 *	Update solid angle
N380 D11 P01 +Q24 P02 +Q5 P03 2 *	Inquire whether an arc is finished. If not finished, return to LBL 2
N390 G11 R+Q6 H+Q5 FQ12 *	Move to the end angle in space
N400 G01 G40 Z+Q23 F1000 *	Retract in the tool axis
N410 G00 G40 X+Q26 *	Pre-position for next arc
N420 D01 Q28 P01 +Q28 P02 +Q18 *	Update rotational position in the plane
N430 D00 Q24 P01 +Q4 *	Reset solid angle
N440 G73 G90 H+Q28 *	Activate new rotational position
N450 D12 P01 +Q28 P02 +Q9 P03 1 *	Unfinished? If not finished, return to label 1
N460 D09 P01 +Q28 P02 +Q9 P03 1 *	
N470 G73 G90 H+0 *	Reset the rotation
N480 G54 X+0 Y+0 Z+0 *	Reset the datum shift
N490 G98 L0 *	End of subprogram
N99999999 %SPHERE G71 *	

9

Programming: Miscellaneous Functions

9.1 Entering Miscellaneous Functions M and STOP

Fundamentals

With the TNC's miscellaneous functions—also called M functions—you can affect

- the program run, e.g., a program interruption
- the machine functions, such as switching spindle rotation and coolant supply on and off
- the path behavior of the tool

The machine tool builder may add some M functions that are not described in this User's Manual. Refer to your machine manual.

You can enter up to two M functions at the end of a positioning block or in a separate block. The TNC displays the following dialog question: **`Miscellaneous function M ?`**

You usually enter only the number of the M function in the programming dialog. Some M functions can be programmed with additional parameters. In this case, the dialog is continued for the parameter input.

In the Manual Operation and Electronic Handwheel modes of operation, the M functions are entered with the M soft key.

Please note that some M functions become effective at the start of a positioning block, and others at the end, regardless of their position in the NC block.

M functions come into effect in the block in which they are called.

Some M functions are effective only in the block in which they are programmed. Unless the M function is only effective blockwise, either you must cancel it in a subsequent block with a separate M function, or it is automatically canceled by the TNC at the end of the program.

Entering an M function in a STOP block

If you program a STOP block, the program run or test run is interrupted at the block, for example for tool inspection. You can also enter an M function in a STOP block:

STOP

- To program an interruption of program run, press the STOP key
- Enter a miscellaneous function M

Example NC blocks

```
N87 G36 M6
```

9.2 Miscellaneous Functions for Program Run Control, Spindle and Coolant

Overview

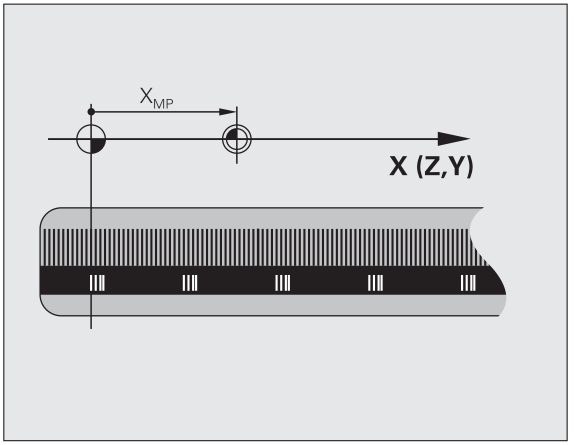
M	Effect	Effective at block...	Start	End
M0	Stop program run Spindle STOP Coolant OFF			■
M1	Optional program STOP Spindle STOP Coolant OFF			■
M2	Stop program run Spindle STOP Coolant OFF Go to block 1 Clear the status display (depends on the **`clearMode`** machine parameter)			■
M3	Spindle ON clockwise		■	
M4	Spindle ON counterclockwise		■	
M5	Spindle STOP			■
M6	Tool change Spindle STOP Program STOP			■
M8	Coolant ON		■	
M9	Coolant OFF			■
M13	Spindle ON clockwise Coolant ON		■	
M14	Spindle ON counterclockwise Coolant ON		■	
M30	Same as M2			■

9.3 Miscellaneous Functions for Coordinate Data

Programming machine-referenced coordinates: M91/M92

Scale reference point

On the scale, a reference mark indicates the position of the scale reference point.



Machine datum

The machine datum is required for the following tasks:

- Defining the limits of traverse (software limit switches)
- Moving to machine-referenced positions (such as tool change positions)
- Setting the workpiece datum

The distance in each axis from the scale reference point to the machine datum is defined by the machine tool builder in a machine parameter.

Standard behavior

The TNC references coordinates to the workpiece datum (see "Datum Setting without a 3-D Touch Probe", page 334).

Behavior with M91—Machine datum

If you want the coordinates in a positioning block to be referenced to the machine datum, end the block with M91.

> If you program incremental coordinates in an M91 block, enter them with respect to the last programmed M91 position. If no M91 position is programmed in the active NC block, then enter the coordinates with respect to the current tool position.

The coordinate values on the TNC screen are referenced to the machine datum. Switch the display of coordinates in the status display to REF (see "Status Displays", page 61).

Behavior with M92—Additional machine datum

In addition to the machine datum, the machine tool builder can also define an additional machine-based position as a reference point.

For each axis, the machine tool builder defines the distance between the machine datum and this additional machine datum. Refer to the machine manual for more information.

If you want the coordinates in a positioning block to be based on the additional machine datum, end the block with M92.

Radius compensation remains the same in blocks that are programmed with M91 or M92. The tool length, however, is **not** compensated.

Effect

M91 and M92 are effective only in the blocks in which they are programmed.

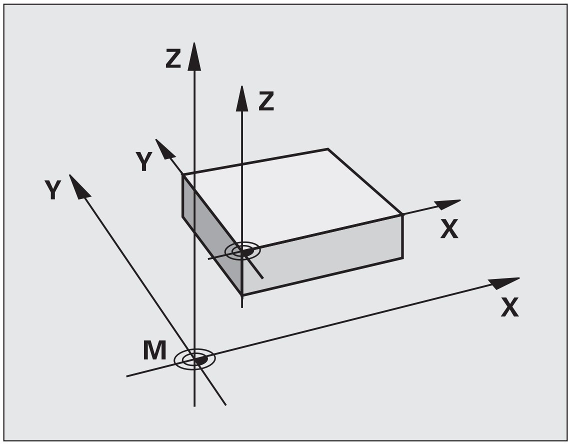
M91 and M92 take effect at the start of block.

Workpiece datum

If you want the coordinates to always be referenced to the machine datum, you can inhibit datum setting for one or more axes.

If datum setting is inhibited for all axes, the TNC no longer displays the SET DATUM soft key in the Manual Operation mode.

The figure shows coordinate systems with the machine datum and workpiece datum.



M91/M92 in the Test Run mode

In order to be able to graphically simulate M91/M92 movements, you need to activate working space monitoring and display the workpiece blank referenced to the set datum (see "Showing the Blank in the Working Space", page 377).

Moving to positions in a non-tilted coordinate system with a tilted working plane: M130

Standard behavior with a tilted working plane

The TNC places the coordinates in the positioning blocks in the tilted coordinate system.

Behavior with M130

The TNC places coordinates in straight line blocks in the untilted coordinate system.

The TNC then positions the (tilted) tool to the programmed coordinates of the untilted system.

Danger of collision!

Subsequent positioning blocks or fixed cycles are carried out in a tilted coordinate system. This can lead to problems in fixed cycles with absolute pre-positioning.

The function M130 is allowed only if the tilted working plane function is active.

Effect

M130 functions blockwise in straight-line blocks without tool radius compensation.

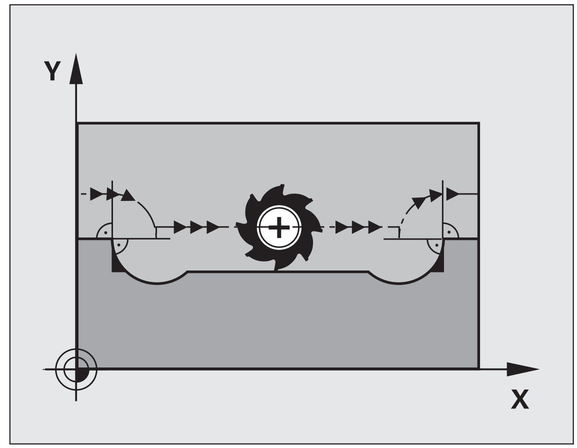
9.4 Miscellaneous Functions for Contouring Behavior

Machining small contour steps: M97

Standard behavior

The TNC inserts a transition arc at outside corners. If the contour steps are very small, however, the tool would damage the contour.

In such cases the TNC interrupts program run and generates the error message "Tool radius too large."

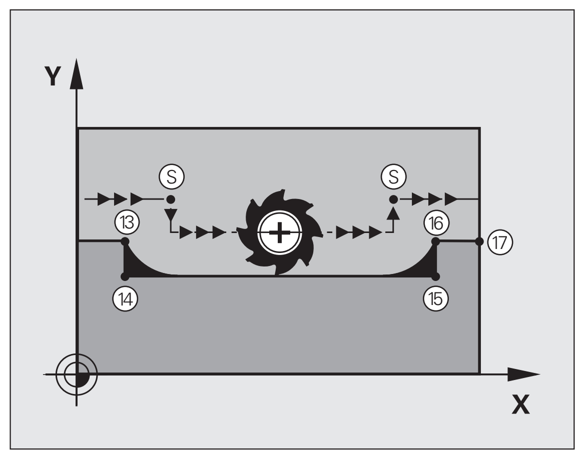


Behavior with M97

The TNC calculates the intersection of the contour elements—as at inside corners—and moves the tool over this point.

Program M97 in the same block as the outside corner.

Instead of **M97** you should use the much more powerful function **M120 LA** (see "Calculating the radius-compensated path in advance (LOOK AHEAD): M120" on page 280).



Effect

M97 is effective only in the blocks in which it is programmed.

A corner machined with M97 will not be completely finished. You may wish to rework the contour with a smaller tool.

Example NC blocks

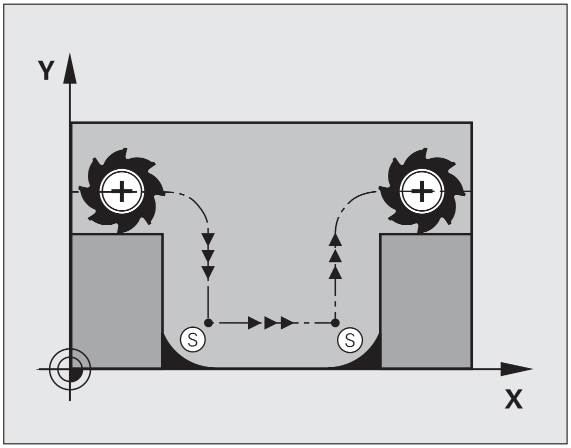
`N50 G99 G01 ... R+20 *`	Large tool radius
`...`	
`N130 X ... Y ... F ... M97 *`	Move to contour point 13
`N140 G91 Y-0.5 ... F ... *`	Machine small contour step 13 to 14
`N150 X+100 ... *`	Move to contour point 15
`N160 Y+0.5 ... F ... M97 *`	Machine small contour step 15 to 16
`N170 G90 X ... Y ... *`	Move to contour point 17

Machining open contours corners: M98

Standard behavior

The TNC calculates the intersections of the cutter paths at inside corners and moves the tool in the new direction at those points.

If the contour is open at the corners, however, this will result in incomplete machining.



Behavior with M98

With the miscellaneous function M98, the TNC temporarily suspends radius compensation to ensure that both corners are completely machined:

Effect

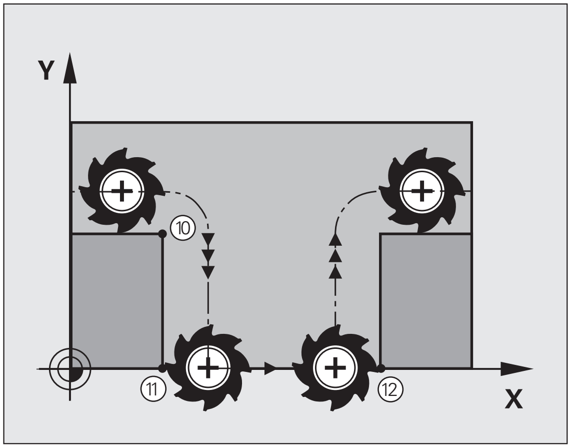
M98 is effective only in the blocks in which it is programmed.

M98 takes effect at the end of block.

Example NC blocks

Move to the contour points 10, 11 and 12 in succession:

```
N100 G01 G41 X ... Y ... F ... *
N110 X ... G91 Y ... M98 *
N120 X+ ... *
```



Feed rate factor for plunging movements: M103

Standard behavior

The TNC moves the tool at the last programmed feed rate, regardless of the direction of traverse.

Behavior with M103

The TNC reduces the feed rate when the tool moves in the negative direction of the tool axis. The feed rate for plunging FZMAX is calculated from the last programmed feed rate FPROG and a factor F%:

FZMAX = FPROG x F%

Programming M103

If you enter M103 in a positioning block, the TNC continues the dialog by asking you the factor F.

Effect

M103 becomes effective at the start of block.
To cancel M103, program M103 once again without a factor.

> M103 is also effective in an active tilted working plane. The feed rate reduction is then effective during traverse in the negative direction of the **tilted** tool axis.

Example NC blocks

The feed rate for plunging is to be 20% of the feed rate in the plane.

`...`	Actual contouring feed rate (mm/min):
`N170 G01 G41 X+20 Y+20 F500 M103 F20 *`	500
`N180 Y+50 *`	500
`N190 G91 Z–2.5 *`	100
`N200 Y+5 Z-5 *`	141
`N210 X+50 *`	500
`N220 G90 Z+5 *`	500

Feed rate in millimeters per spindle revolution: M136

Standard behavior

The TNC moves the tool at the programmed feed rate F in mm/min.

Behavior with M136

> In inch-programs, M136 is not permitted in combination with the new alternate feed rate FU.
>
> The spindle is not permitted to be controlled when M136 is active.

With M136, the TNC does not move the tool in mm/min, but rather at the programmed feed rate F in millimeters per spindle revolution. If you change the spindle speed by using the spindle override, the TNC changes the feed rate accordingly.

Effect

M136 becomes effective at the start of block.

You can cancel M136 by programming M137.

Feed rate for circular arcs: M109/M110/M111

Standard behavior

The TNC applies the programmed feed rate to the path of the tool center.

Behavior at circular arcs with M109

The TNC adjusts the feed rate for circular arcs at inside and outside contours so that the feed rate at the tool cutting edge remains constant.

Behavior at circular arcs with M110

The TNC keeps the feed rate constant for circular arcs at inside contours only. At outside contours, the feed rate is not adjusted.

> If you define M109 or M110 before calling a machining cycle with a number greater than 200, the adjusted feed rate is also effective for circular arcs within these machining cycles. The initial state is restored after finishing or aborting a machining cycle.

Effect

M109 and M110 become effective at the start of block. To cancel M109 or M110, enter M111.

Calculating the radius-compensated path in advance (LOOK AHEAD): M120

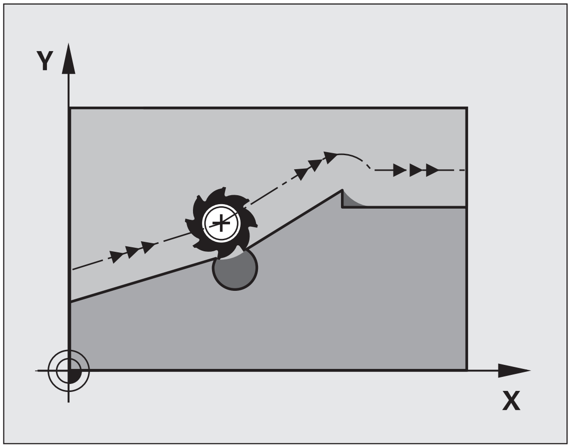
Standard behavior

If the tool radius is larger than the contour step that is to be machined with radius compensation, the TNC interrupts program run and generates an error message. M97 (see "Machining small contour steps: M97" on page 275) inhibits the error message, but this results in dwell marks and will also move the corner.

If the programmed contour contains undercut features, the tool may damage the contour.

Behavior with M120

The TNC checks radius-compensated paths for contour undercuts and tool path intersections, and calculates the tool path in advance from the current block. Areas of the contour that might be damaged by the tool are not machined (dark areas in figure). You can also use M120 to calculate the radius compensation for digitized data or data created on an external programming system. This means that deviations from the theoretical tool radius can be compensated.



Use LA (Look Ahead) after M120 to define the number of blocks (maximum: 99) that you want the TNC to calculate in advance. Note that the larger the number of blocks you choose, the higher the block processing time will be.

Input

If you enter M120 in a positioning block, the TNC continues the dialog for this block by asking you the number of blocks LA that are to be calculated in advance.

Effect

M120 must be located in an NC block that also contains radius compensation **G41** or **G42**. M120 is then effective from this block until

- radius compensation is canceled with **G40**, or
- M120 LA0 is programmed, or
- M120 is programmed without LA, or
- another program is called with **%**, or
- the working plane is tilted with Cycle **G80** or the PLANE function.

M120 becomes effective at the start of block.

Restrictions

- After an external or internal stop, you can only re-enter the contour with the function RESTORE POS. AT N. Before you start the block scan, you must cancel M120, otherwise the TNC will output an error message.
- When using the path functions **G25** and **G24**, the blocks before and after **G25** or **G24** must contain only coordinates in the working plane.
- Before using the functions listed below, you have to cancel M120 and the radius compensation:
  - Cycle **G60** Tolerance
  - Cycle **G80** Working plane
  - PLANE function
  - M114
  - M128

Superimposing handwheel positioning during program run: M118

Standard behavior

In the program run modes, the TNC moves the tool as defined in the part program.

Behavior with M118

M118 permits manual corrections by handwheel during program run. Just program M118 and enter an axis-specific value (linear or rotary axis) in millimeters.

Input

If you enter M118 in a positioning block, the TNC continues the dialog for this block by asking you the axis-specific values. The coordinates are entered with the orange axis direction buttons or the ASCII keyboard.

Effect

Cancel handwheel positioning by programming M118 once again without coordinate input.

M118 becomes effective at the start of block.

Example NC blocks

You want to be able to use the handwheel during program run to move the tool in the working plane X/Y by ±1 mm and in the rotary axis B by ±5° from the programmed value:

```
N250 G01 G41 X+0 Y+38.5 F125 M118 X1 Y1 B5 *
```

M118 is effective in a tilted coordinate system if you activate the tilted working plane function for the Manual Operation mode. If the tilted working plane function is not active for the Manual Operation mode, the original coordinate system is effective.

M118 also functions in the Positioning with MDI mode of operation!

If M118 is active, the MANUAL TRAVERSE function is not available after a program interruption!

Retraction from the contour in the tool-axis direction: M140

Standard behavior

In the program run modes, the TNC moves the tool as defined in the part program.

Behavior with M140

With M140 MB (move back) you can enter a path in the direction of the tool axis for departure from the contour.

Input

If you enter M140 in a positioning block, the TNC continues the dialog and asks for the desired path of tool departure from the contour. Enter the requested path that the tool should follow when departing the contour, or press the MB MAX soft key to move to the limit of the traverse range.

In addition, you can program the feed rate at which the tool traverses the entered path. If you do not enter a feed rate, the TNC moves the tool along the entered path at rapid traverse.

Effect

M140 is effective only in the block in which it is programmed.

M140 becomes effective at the start of block.

Example NC blocks

Block 250: Retract the tool 50 mm from the contour.

Block 251: Move the tool to the limit of the traverse range.

```
N250 G01 X+0 Y+38.5 F125 M140 MB50 *
N251 G01 X+0 Y+38.5 F125 M140 MB MAX *
```

M140 is also effective if the tilted-working-plane function is active. On machines with tilting heads, the TNC then moves the tool in the tilted coordinate system.

With **M140 MB MAX** you can only retract in positive direction.

Always define a TOOL CALL with a tool axis before entering **M140,** otherwise the direction of traverse is not defined.

Suppressing touch probe monitoring: M141

Standard behavior

When the stylus is deflected, the TNC outputs an error message as soon as you attempt to move a machine axis.

Behavior with M141

The TNC moves the machine axes even if the touch probe is deflected. This function is required if you wish to write your own measuring cycle in connection with measuring cycle 3 in order to retract the stylus by means of a positioning block after it has been deflected.

Danger of collision!

If you use M141, make sure that you retract the touch probe in the correct direction.

M141 functions only for movements with straight-line blocks.

Effect

M141 is effective only in the block in which it is programmed.

M141 becomes effective at the start of the block.

Automatically retract tool from the contour at an NC stop: M148

Standard behavior

At an NC stop the TNC stops all traverse movements. The tool stops moving at the point of interruption.

Behavior with M148

The M148 function must be enabled by the machine tool builder. The machine tool builder defines in a machine parameter the path that the TNC is to traverse for a **LIFTOFF** command.

The TNC retracts the tool by up to 2 mm in the direction of the tool axis if, in the **LIFTOFF** column of the tool table, you set the parameter **Y** for the active tool (see "Tool table: Standard tool data" on page 134).

LIFTOFF takes effect in the following situations:

- An NC stop triggered by you
- An NC stop triggered by the software, e.g. if an error occurred in the drive system
- When a power interruption occurs

Danger of collision!

Remember that, especially on curved surfaces, the surface can be damaged during return to the contour. Back the tool off before returning to the contour!

In the **CfgLiftOff** machine parameter, define the value by which the tool is to be retracted. In the **CfgLiftOff** machine parameter you can also switch off the function.

Effect

M148 remains in effect until deactivated with M149.

M148 becomes effective at the start of block, M149 at the end of block.

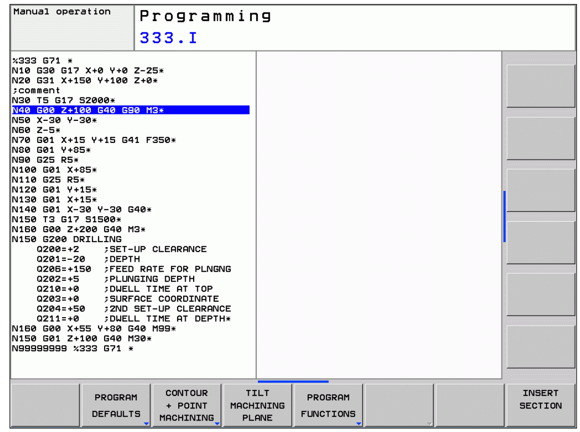
10

Programming: Special Functions

10.1 Overview of Special Functions

Press the SPEC FCT and the corresponding soft keys to access further special functions of the TNC. The following tables will give you an overview of which functions are available.

Main menu for SPEC FCT special functions



SPEC FCT

▶ Press the special functions key

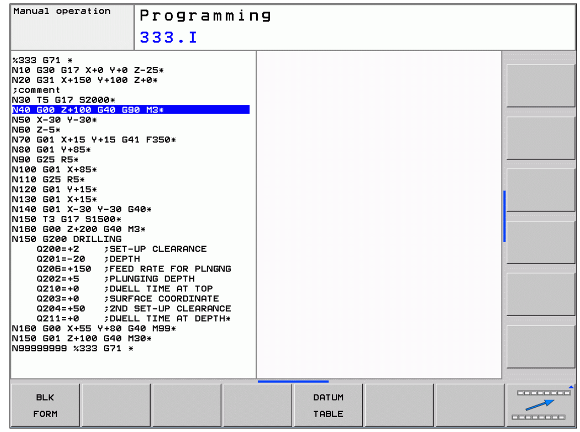
Function	Soft key	Description
Define program defaults	PROGRAM DEFAULTS	Page 289
Functions for contour and point machining	CONTOUR + POINT MACHINING	Page 289
Define the **`PLANE`** function	TILT MACHINING PLANE	Page 301
Define different DIN/ISO functions	PROGRAM FUNCTIONS	Page 290
Define structure items	INSERT SECTION	Page 113

Program defaults menu

PROGRAM DEFAULTS

▶ Select the program defaults menu

Function	Soft key	Description
Define the workpiece blank	BLK FORM	Page 79
Select datum table	DATUM TABLE	See User's Manual for Cycles

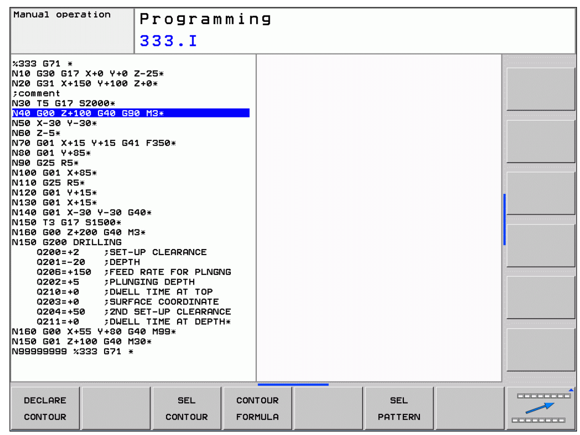


Functions for contour and point machining menu

CONTOUR + POINT MACHINING

▶ Select the menu for functions for contour and point machining.

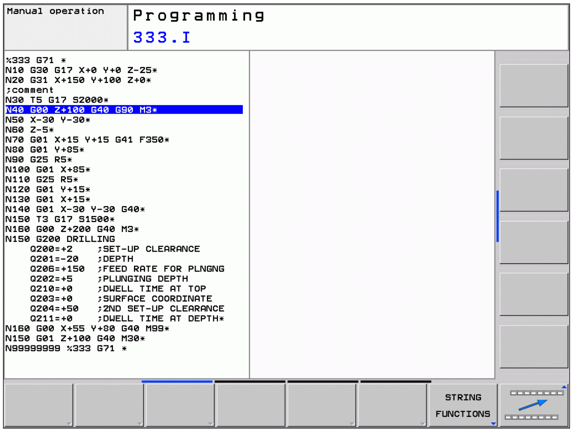
Function	Soft key	Description
Assign contour description	DECLARE CONTOUR	See User's Manual for Cycles
Select a contour definition	SEL CONTOUR	See User's Manual for Cycles
Define a complex contour formula	CONTOUR FORMULA	See User's Manual for Cycles



Menu of various DIN/ISO functions

PROGRAM FUNCTIONS

▶ Select the menu for defining various DIN/ISO functions



Function	Soft key	Description
Define string functions	STRING FUNCTIONS	Page 245
Defining DIN/ISO Functions	DIN/ISO	Page 291
Adding comments	INSERT COMMENT	Page 111

10.2 Defining DIN/ISO Functions

Overview

If a USB keyboard is connected, you can also enter the DIN/ISO functions by using the USB keyboard.

The TNC provides soft keys with the following functions for creating DIN/ISO programs:

Function	Soft key
Select DIN/ISO functions	DIN/ISO
Feed rate	F
Tool movements, cycles and program functions	G
X coordinate of the circle center/pole	I
Y coordinate of the circle center/pole	J
Label call for subprogram and program section repeat	L
Miscellaneous function	M
Block number	N
Tool call	T
Polar coordinate angle	H
Z coordinate of the circle center/pole	K
Polar coordinate radius	R
Spindle speed	S

10.3 Creating Text Files

Application

You can use the TNC's text editor to write and edit texts. To do so, connect a USB keyboard to the TNC. Typical applications:

- Recording test results
- Documenting working procedures
- Creating formula collections

Text files have the extension .A (for ASCII files). If you want to edit other types of files, you must first convert them into type .A files.

Opening and exiting text files

- Select the Programming and Editing mode of operation
- Press the PGM MGT key to call the file manager
- To display type .A files, press the SELECT TYPE and then the SHOW .A soft keys.
- Select a file and open it with the SELECT soft key or ENT key, or create a new file by entering the new file name and confirming your entry with the ENT key.

To leave the text editor, call the file manager and select a file of a different file type, for example a part program.

Cursor movements	Soft key
Move cursor one word to the right	MOVE WORD →
Move cursor one word to the left	MOVE WORD ←
Go to next screen page	PAGE ↓
Go to previous screen page	PAGE ↑
Go to beginning of file	BEGIN ↑
Go to end of file	END ↓

Editing texts

Above the first line of the text editor, there is an information field showing the file name, location and line information:

`File:`	Name of the text file
`Line:`	Line in which the cursor is presently located
`Column:`	Column in which the cursor is presently located

The text is inserted or overwritten at the location of the cursor. You can move the cursor to any desired position in the text file by pressing the arrow keys.

The line in which the cursor is presently located is depicted in a different color. You can insert a line break with the Return or ENT key.

Deleting and inserting characters, words and lines

With the text editor, you can erase words and even lines, and insert them at any desired location in the text.

- Move the cursor to the word or line that you wish to erase and insert at a different place in the text
- Press the DELETE WORD or DELETE LINE soft key. The text is placed in the buffer memory
- Move the cursor to the location where you wish to insert the text, and press the RESTORE LINE/WORD soft key

Function	Soft key
Delete and temporarily store a line	DELETE LINE
Delete and temporarily store a word	DELETE WORD
Delete and temporarily store a character	DELETE CHAR
Insert a line or word from temporary storage	INSERT LINE / WORD

Editing text blocks

You can copy and erase text blocks of any size, and insert them at other locations. Before any of these actions, you must first select the desired text block:

- To select a text block, move the cursor to the first character of the text you wish to select

SELECT BLOCK

- Press the SELECT BLOCK soft key
- Move the cursor to the last character of the text you wish to select You can select whole lines by moving the cursor up or down directly with the arrow keys—the selected text is shown in a different color

After selecting the desired text block, you can edit the text with the following soft keys:

Function	Soft key
Delete the selected text and store temporarily	CUT OUT BLOCK
Store marked block temporarily without erasing (copy)	INSERT BLOCK

If desired, you can now insert the temporarily stored block at a different location:

- Move the cursor to the location where you want to insert the temporarily stored text block

INSERT BLOCK

- Press the INSERT BLOCK soft key for the text block to be inserted

You can insert the temporarily stored text block as often as desired

To transfer the selected text to a different file

- Select the text block as described previously

APPEND TO FILE

- Press the APPEND TO FILE soft key. The TNC displays the dialog prompt **`Destination file =`**
- Enter the path and name of the destination file. The TNC appends the selected text to the specified file. If no target file with the specified name is found, the TNC creates a new file with the selected text.

To insert another file at the cursor position,

- Move the cursor to the location in the text where you wish to insert another file

READ FILE

- Press the READ FILE soft key. The TNC displays the dialog prompt **`File name =`**
- Enter the path and name of the file you want to insert

Finding text sections

With the text editor, you can search for words or character strings in a text. Two functions are available:

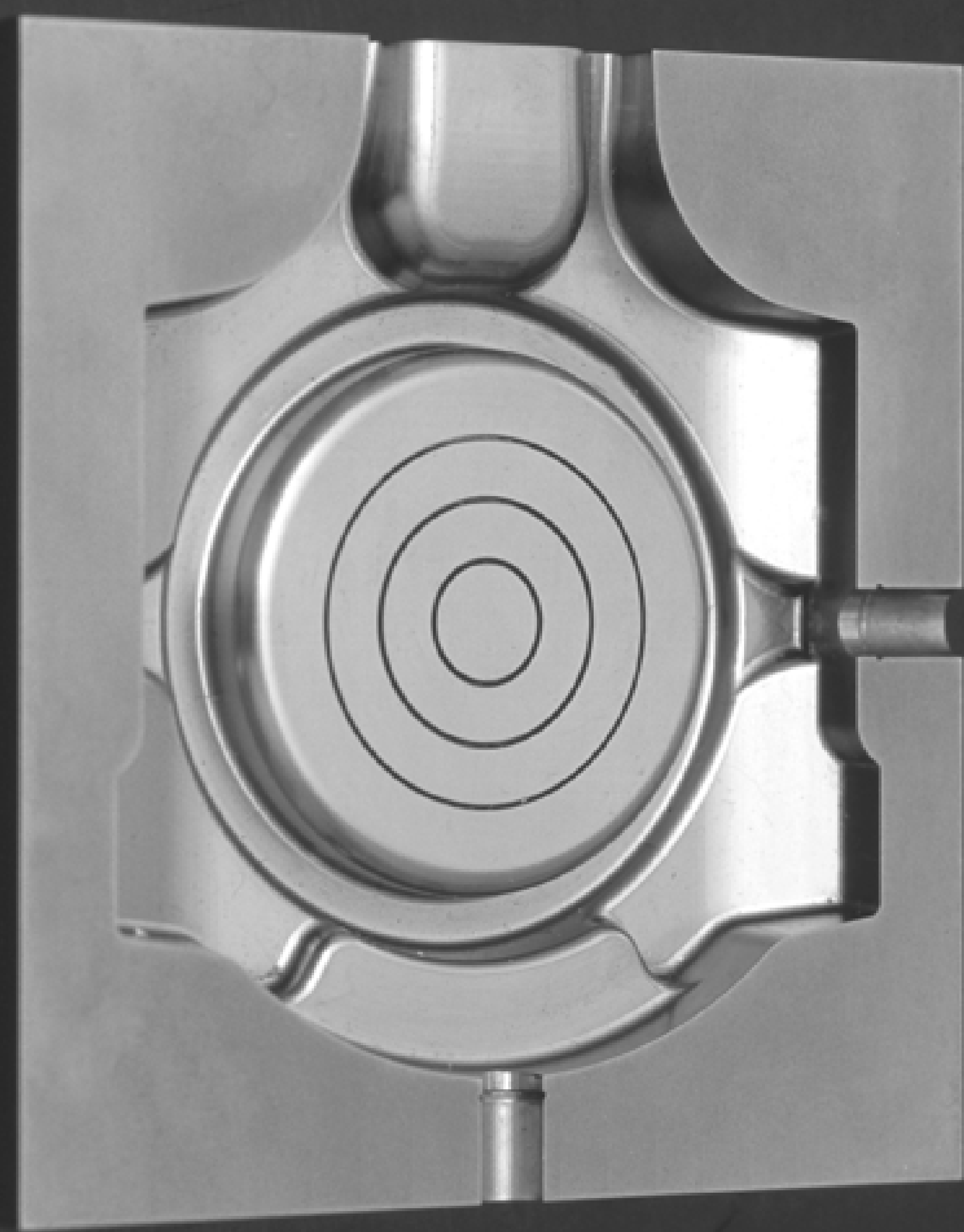
Finding the current text

The search function is used for finding the next occurrence of the word in which the cursor is presently located:

- Move the cursor to the desired word.
- To select the search function, press the FIND soft key.
- Press the FIND CURRENT WORD soft key.
- To leave the search function, press the END soft key.

Finding any text

- To select the search function, press the FIND soft key. The TNC displays the dialog prompt **`Find text:`**
- Enter the text that you wish to find
- To find the text, press the EXECUTE soft key
- To leave the search function, press the END soft key.



11

Programming: Multiple Axis Machining

11.1 Functions for Multiple Axis Machining

The TNC functions for multiple axis machining are described in this chapter.

TNC function	Description	Page
PLANE	Define machining in the tilted working plane	Page 299
M116	Feed rate of rotary axes	Page 320
M126	Shortest-path traverse of rotary axes	Page 321
M94	Reduce display value of rotary axes	Page 322
M138	Selection of tilted axes	Page 323
M144	Calculate machine kinematics	Page 324

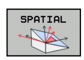
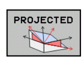
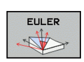
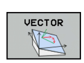
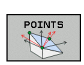
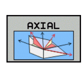
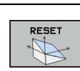
11.2 The PLANE Function: Tilting the Working Plane (Software Option 1)

Introduction

> The machine manufacturer must enable the functions for tilting the working plane!
>
> You can only use the **PLANE** function in its entirety on machines which have at least two rotary axes (head and/or table). Exception: **PLANE AXIAL** can also be used if only a single rotary axis is present or active on your machine.

The **PLANE** function is a powerful function for defining tilted working planes in various manners.

All **PLANE** functions available on the TNC describe the desired working plane independently of the rotary axes actually present on your machine. The following possibilities are available:

Function	Required parameters	Soft key	Page
SPATIAL	Three space angles: **SPA, SPB,** and **SPC**		Page 303
PROJECTED	Two projection angles: **PROPR** and **PROMIN** and a rotation angle **ROT**		Page 305
EULER	Three Euler angles: precession **(EULPR),** nutation **(EULNU)** and rotation **(EULROT)**		Page 307
VECTOR	Norm vector for defining the plane and base vector for defining the direction of the tilted X axis		Page 309
POINTS	Coordinates of any three points in the plane to be tilted		Page 311
RELATIVE	Single, incrementally effective spatial angle		Page 313
AXIAL	Up to three absolute or incremental axis angles **A, B, C**		Page 314
RESET	Reset the PLANE function		Page 302

The parameter definition of the **PLANE** function is separated into two parts:

- The geometric definition of the plane, which is different for each of the available **PLANE** functions.
- The positioning behavior of the **PLANE** function, which is independent of the plane definition and is identical for all **PLANE** functions (see "Specifying the positioning behavior of the PLANE function" on page 316).

The actual-position-capture function is not possible with an active tilted working plane.

If you use the **PLANE** function when **M120** is active, the TNC automatically rescinds the radius compensation, which also rescinds the **M120** function.

Always use **PLANE RESET** to reset **PLANE** functions. Entering 0 in all **PLANE** parameters does not completely reset the function.

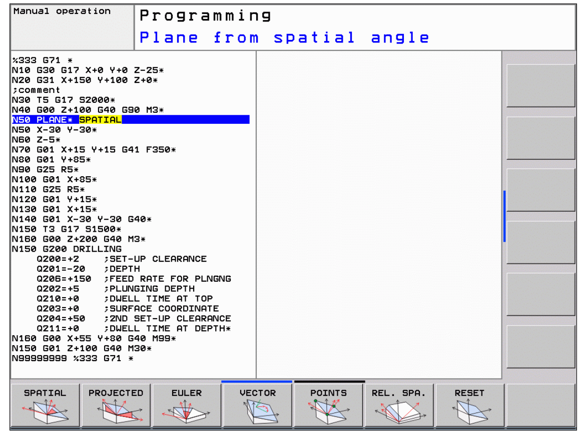
Define the PLANE function

SPEC FCT

▶ Show the soft-key row with special functions

TILT MACHINING PLANE

▶ Select the **PLANE** function: Press the TILT MACHINING PLANE soft key: The TNC displays the available definition possibilities in the soft-key row

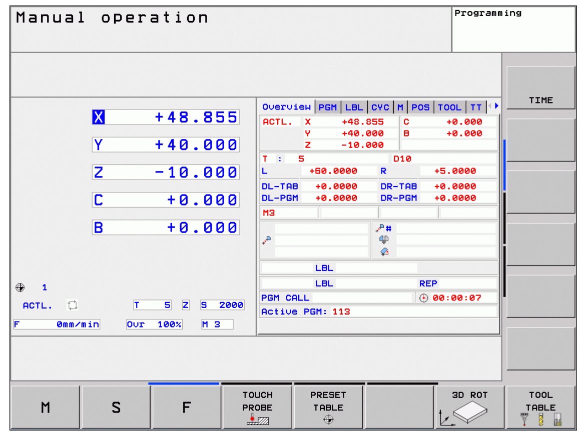


Selecting functions

▶ Select the desired function by soft key. The TNC continues the dialog and requests the required parameters

Position display

As soon as a **PLANE** function is active, the TNC shows the calculated spatial angle in the additional status display (see figure). As a rule, the TNC internally always calculates with space angles, regardless of which **PLANE** function is active.



Reset the PLANE function

SPEC FCT

▶ Show the soft-key row with special functions

SPECIAL TNC FUNCTIONS

▶ Select special TNC functions: Press the SPECIAL TNC FUNCTIONS soft key

TILT MACHINING PLANE

▶ Select the PLANE function: Press the TILT MACHINING PLANE soft key: The TNC displays the available definition possibilities in the soft-key row

RESET

▶ Select the Reset function. This internally resets the **PLANE** function, but does not change the current axis positions

MOVE

▶ Specify whether the TNC should automatically move the rotary axes to the default setting (**MOVE** or **TURN**) or not (**STAY**) (see "Automatic positioning: MOVE/TURN/STAY (entry is mandatory)" on page 316).

END

▶ To terminate entry, press the END key

The **PLANE RESET** function resets the current **PLANE** function—or an active cycle **G80**—completely (angles = 0 and function is inactive). It does not need to be defined more than once.

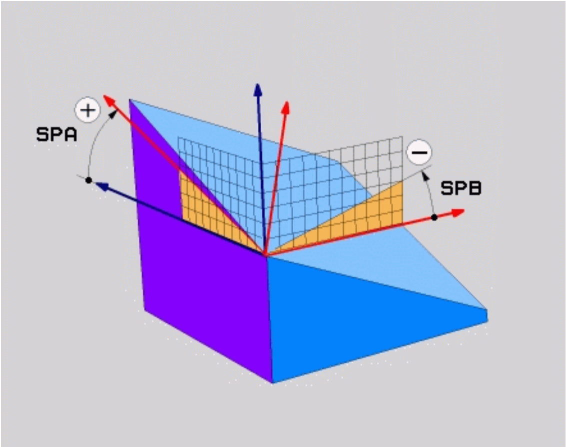
Example: NC block

```
25 PLANE RESET MOVE SET-UP50 F1000
```

Defining the machining plane with space angles: PLANE SPATIAL

Application

Spatial angles define a working plane through up to three **rotations around the fixed machine coordinate system.** The sequence of rotations is firmly specified: first around the A axis, then B, and then C (the function corresponds to Cycle 19, if the entries in Cycle 19 are set to space angles).



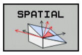
Before programming, note the following

You must always define the three space angles **SPA, SPB,** and **SPC,** even if one of them = 0.

The sequence of the rotations described above is independent of the active tool axis.

Parameter description for the positioning behavior: See "Specifying the positioning behavior of the PLANE function" on page 316.

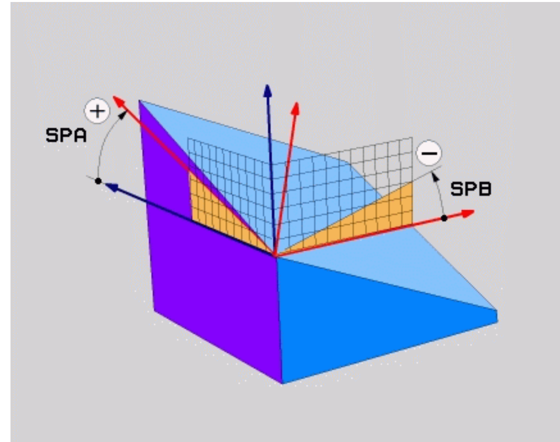
Input parameters

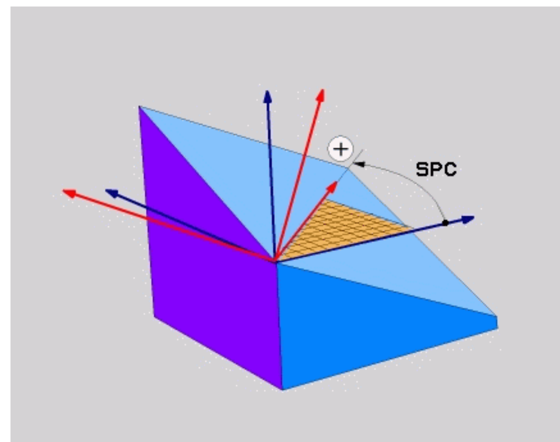


- **`Spatial angle A?`**: Rotational angle **`SPA`** around the fixed machine axis X (see figure at top right). Input range from -359.9999° to +359.9999°
- **`Spatial angle B?:`** Rotational angle **`SPB`** around the fixed machine axis Y (see figure at top right). Input range from -359.9999° to +359.9999°
- **`Spatial angle C?:`** Rotational angle **`SPC`** around the fixed machine axis Z (see figure at center right). Input range from -359.9999° to +359.9999°
- Continue with the positioning properties (see "Specifying the positioning behavior of the PLANE function" on page 316)

Abbreviations used

Abbreviation	Meaning
`SPATIAL`	**Spatial** = in space
`SPA`	Spatial **A**: Rotation about the X axis
`SPB`	Spatial **B**: Rotation about the Y axis
`SPC`	Spatial **C**: rotation about the Z axis





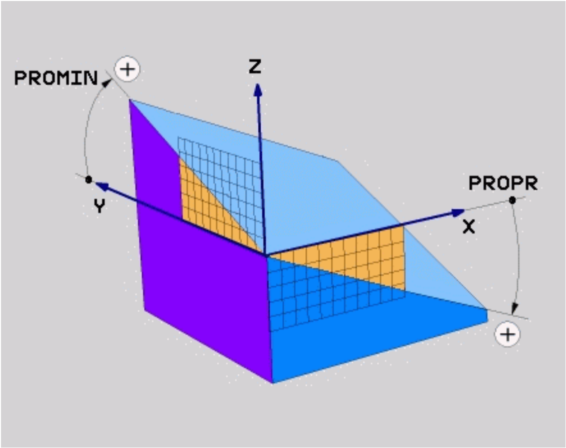
Example: NC block

```
5 PLANE SPATIAL SPA+27 SPB+0 SPC+45 .....
```

Defining the machining plane with projection angles: PROJECTED PLANE

Application

Projection angles define a machining plane through the entry of two angles that you determine by projecting the first coordinate plane (Z/X plane with tool axis Z) and the second coordinate plane (Y/Z with tool axis Z) onto the machining plane to be defined.

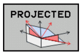


Before programming, note the following

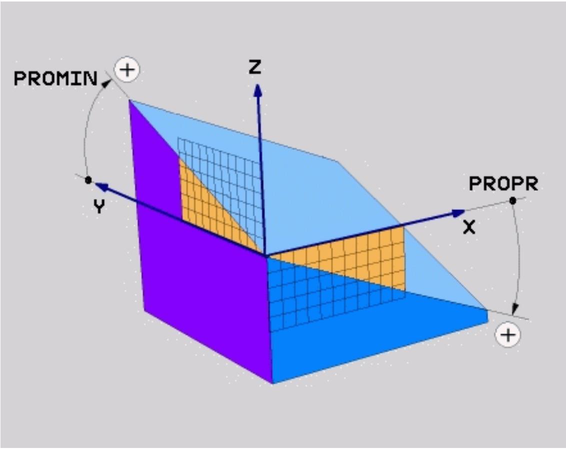
You can only use projection angles if the angle definitions are given with respect to a rectangular cuboid. Otherwise distortions could occur on the workpiece.

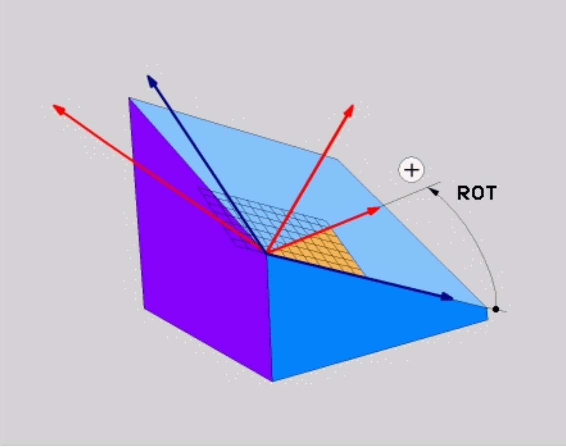
Parameter description for the positioning behavior: See "Specifying the positioning behavior of the PLANE function" on page 316.

Input parameters



- **`Proj. angle 1st coordinate plane?:`** Projected angle of the tilted machining plane in the 1st coordinate plane of the fixed machine coordinate system (Z/X for tool axis Z, see figure at top right). Input range: from -89.9999° to +89.9999°. The 0° axis is the principal axis of the active working plane (X for tool axis Z. See figure at top right for positive direction)
- **`Proj. angle 2nd coordinate plane?:`** Projected angle in the 2nd coordinate plane of the fixed machine coordinate system (Y/Z for tool axis Z, see figure at top right). Input range: from -89.9999° to +89.9999°. The 0° axis is the minor axis of the active machining plane (Y for tool axis Z)
- **`ROT angle of the tilted plane?:`** Rotation of the tilted coordinate system around the tilted tool axis (corresponds to a rotation with Cycle 10 ROTATION). The rotation angle is used to simply specify the direction of the principal axis of the working plane (X for tool axis Z, Z for tool axis Y; see figure at bottom right). Input range: –360° to +360°
- Continue with the positioning properties (see "Specifying the positioning behavior of the PLANE function" on page 316)





NC block

```
5 PLANE PROJECTED PROPR+24 PROMIN+24 PROROT+30 .....
```

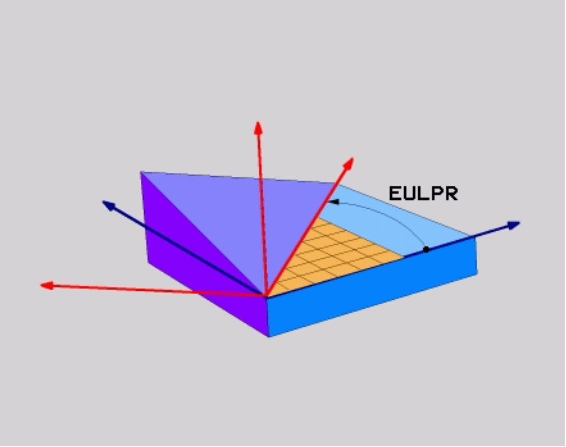
Abbreviations used

Abbreviation	Meaning
PROJECTED	**Projected**
PROPR	Principal plane
PROMIN	Minor plane
PROROT	Rotation

Defining the machining plane with Euler angles: EULER PLANE

Application

Euler angles define a machining plane through up to three **rotations about the respectively tilted coordinate system.** The Swiss mathematician Leonhard Euler defined these angles. When applied to the machine coordinate system, they have the following meanings:



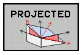
Precession angle **EULPR**	Rotation of the coordinate system around the Z axis
Nutation angle **EULNU**	Rotation of the coordinate system around the X axis already shifted by the precession angle
Rotation angle **EULROT**	Rotation of the tilted machining plane around the tilted Z axis

Before programming, note the following

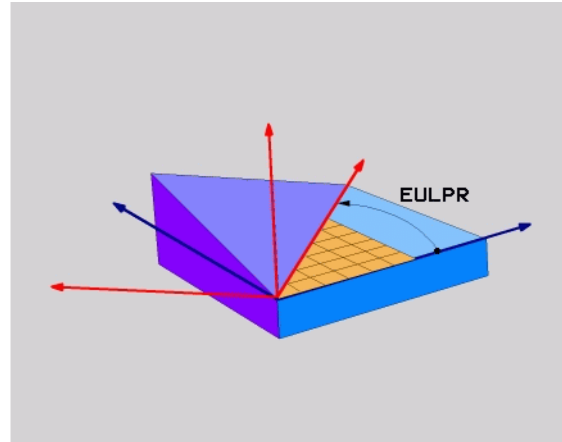
The sequence of the rotations described above is independent of the active tool axis.

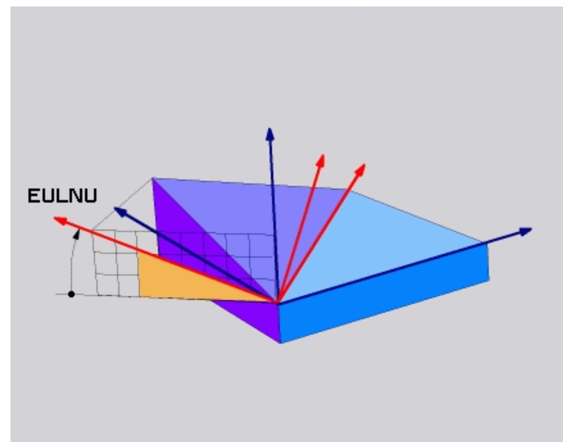
Parameter description for the positioning behavior: See "Specifying the positioning behavior of the PLANE function" on page 316.

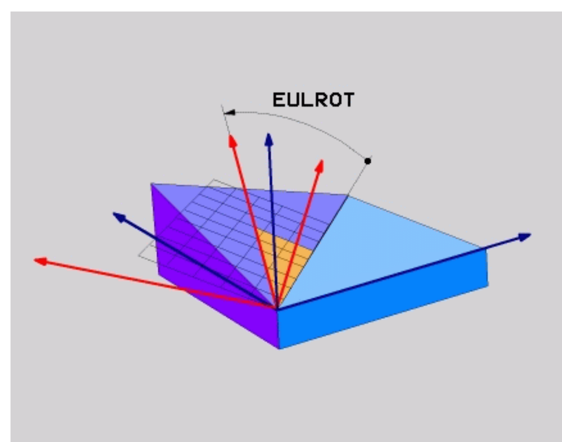
Input parameters



- **`Rot. angle main coordinate plane?:`** Rotary angle **`EULPR`** around the Z axis (see figure at top right). Please note:
  - Input range: –180.0000° to +180.0000°
  - The 0° axis is the X axis
- **`Tilting angle tool axis?:`** Tilting angle **`EULNUT`** of the coordinate system around the X axis shifted by the precession angle (see figure at center right). Please note:
  - Input range: 0° to +180.0000°
  - The 0° axis is the Z axis
- **`ROT angle of the tilted plane?`**: Rotation **`EULROT`** of the tilted coordinate system around the tilted Z axis (corresponds to a rotation with Cycle 10 ROTATION). Use the rotation angle to simply define the direction of the X axis in the tilted machining plane (see figure at bottom right). Please note:
  - Input range: 0° to 360.0000°
  - The 0° axis is the X axis
- Continue with the positioning properties (see "Specifying the positioning behavior of the PLANE function" on page 316)







NC block

```
5 PLANE EULER EULPR45 EULNU20 EULROT22 .....
```

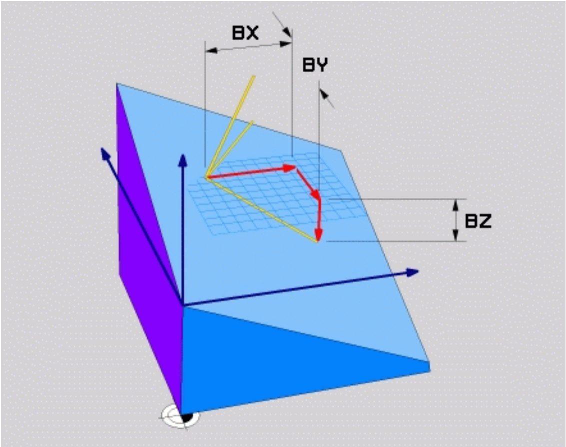
Abbreviations used

Abbreviation	Meaning
`EULER`	Swiss mathematician who defined these angles
`EULPR`	Precession angle: angle describing the rotation of the coordinate system around the Z axis
`EULNU`	Nutation angle: angle describing the rotation of the coordinate system around the X axis shifted by the precession angle
`EULROT`	Rotation angle: angle describing the rotation of the tilted machining plane around the tilted Z axis

Defining the working plane with two vectors: VECTOR PLANE

Application

You can use the definition of a working plane via **two vectors** if your CAD system can calculate the base vector and normal vector of the tilted machining plane. A normalized input is not necessary. The TNC calculates the normal, so you can enter values between -9.999999 and +9.999999.



The base vector required for the definition of the machining plane is defined by the components **BX, BY** and **BZ** (see figure at right). The normal vector is defined by the components **NX, NY** and **NZ.**

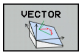
The base vector defines the direction of the X axis in the tilted working plane, and the normal vector determines the direction of the tool axis, and at the same time is perpendicular to it.

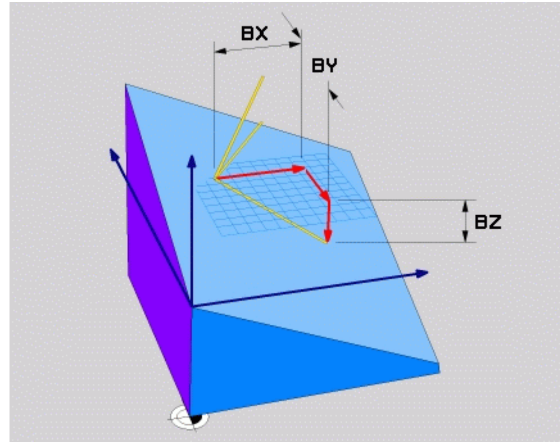
Before programming, note the following

The TNC calculates standardized vectors from the values you enter.

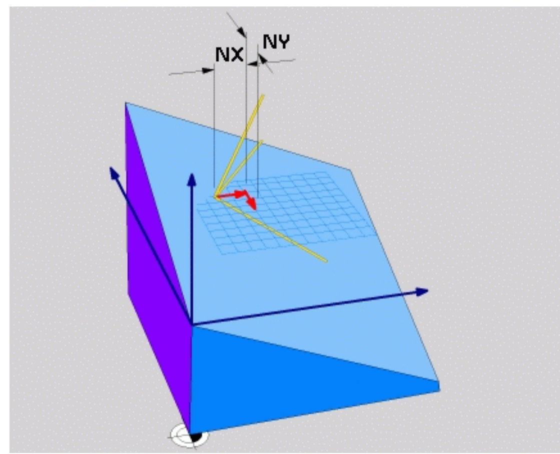
Parameter description for the positioning behavior: See "Specifying the positioning behavior of the PLANE function" on page 316.

Input parameters



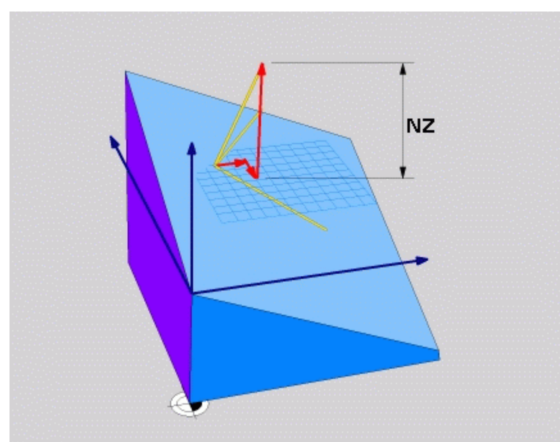


- **`X component of base vector?:`** X component **`BX`** of the base vector B (see figure at top right). Input range: –9.9999999 to +9.9999999
- **`Y component of base vector?:`** Y component **`BY`** of the base vector B (see figure at top right). Input range: –9.9999999 to +9.9999999
- **`Z component of base vector?:`** Z component **`BZ`** of the base vector B (see figure at top right). Input range: –9.9999999 to +9.9999999
- **`X component of normal vector?:`** X component **`NX`** of the normal vector N (see figure at center right). Input range: –9.9999999 to +9.9999999
- **`Y component of normal vector?:`** Y component **`NY`** of the normal vector N (see figure at center right). Input range: –9.9999999 to +9.9999999
- **`Z component of normal vector?:`** Z component **`NZ`** of the normal vector N (see figure at lower right). Input range: –9.9999999 to +9.9999999
- Continue with the positioning properties (see "Specifying the positioning behavior of the PLANE function" on page 316)



NC block

```
5 PLANE VECTOR BX0.8 BY-0.4 BZ-
0.4472 NX0.2 NY0.2 NZ0.9592 ...
```



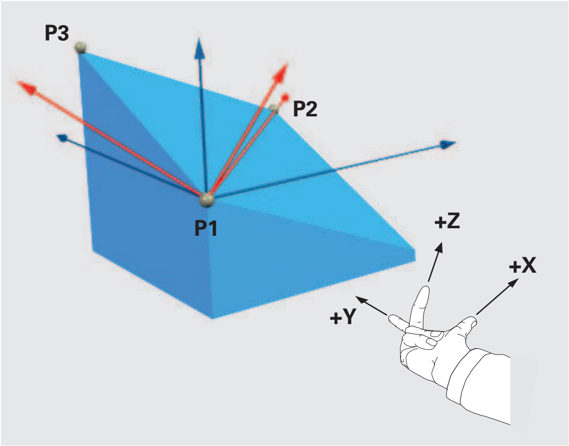
Abbreviations used

Abbreviation	Meaning
`VECTOR`	Vector
`BX, BY, BZ`	Base vector: **X, Y** and **Z** components
`NX, NY, NZ`	Normal vector: **X, Y** and **Z** components

Defining the working plane via three points: PLANE POINTS

Application

A machining plane can be uniquely defined by entering **any three points P1 to P3 in this plane.** This possibility is realized in the **PLANE POINTS** function.



Before programming, note the following

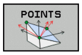
The connection from Point 1 to Point 2 determines the direction of the tilted main axis (X for tool axis Z).

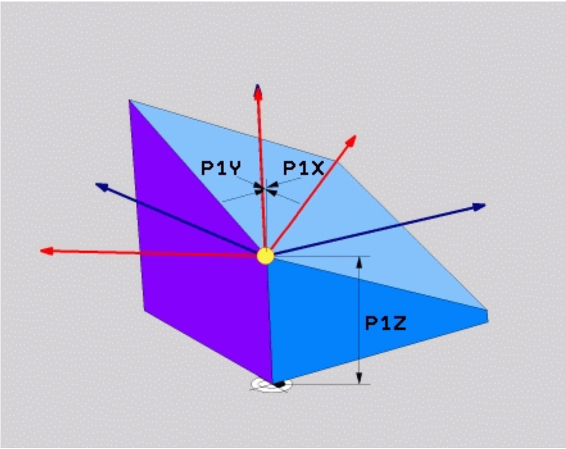
The direction of the tilted tool axis is determined by the position of Point 3 relative to the connecting line between Point 1 and Point 2. Use the right-hand rule (thumb = X axis, index finger = Y axis, middle finger = Z axis (see figure at right)) to remember: thumb (X axis) points from Point 1 to Point 2, index finger (Y axis) points parallel to the tilted Y axis in the direction of Point 3. Then the middle finger points in the direction of the tilted tool axis.

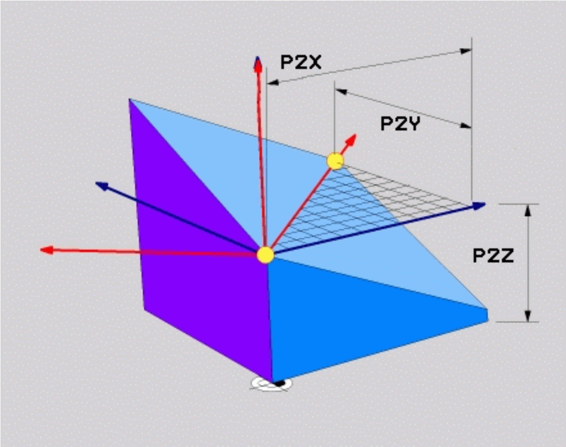
The three points define the slope of the plane. The position of the active datum is not changed by the TNC.

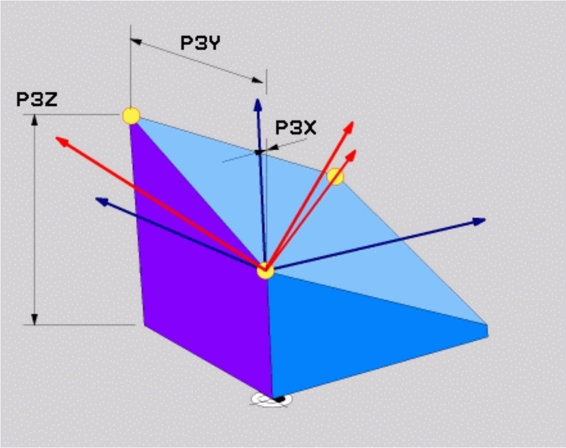
Parameter description for the positioning behavior: See "Specifying the positioning behavior of the PLANE function" on page 316.

Input parameters









- **`X coordinate of 1st plane point?`**: X coordinate **`P1X`** of the 1st plane point (see figure at top right)
- **`Y coordinate of 1st plane point?`**: Y coordinate **`P1Y`** of the 1st plane point (see figure at top right)
- **`Z coordinate of 1st plane point?`**: Z coordinate **`P1Z`** of the 1st plane point (see figure at top right)
- **`X coordinate of 2nd plane point?`**: X coordinate **`P2X`** of the 2nd plane point (see figure at center right)
- **`Y coordinate of 2nd plane point?`**: Y coordinate **`P2Y`** of the 2nd plane point (see figure at center right)
- **`Z coordinate of 2nd plane point?`**: Z coordinate **`P2Z`** of the 2nd plane point (see figure at center right)
- **`X coordinate of 3rd plane point?`**: X coordinate **`P3X`** of the 3rd plane point (see figure at bottom right)
- **`Y coordinate of 3rd plane point?`**: Y coordinate **`P3Y`** of the 3rd plane point (see figure at bottom right)
- **`Z coordinate of 3rd plane point?`**: Z coordinate **`P3Z`** of the 3rd plane point (see figure at bottom right)
- Continue with the positioning properties (see "Specifying the positioning behavior of the PLANE function" on page 316)

NC block

```
5 PLANE POINTS P1X+0 P1Y+0 P1Z+20 P2X+30 P2Y+31 P2Z+20
P3X+0 P3Y+41 P3Z+32.5 .....
```

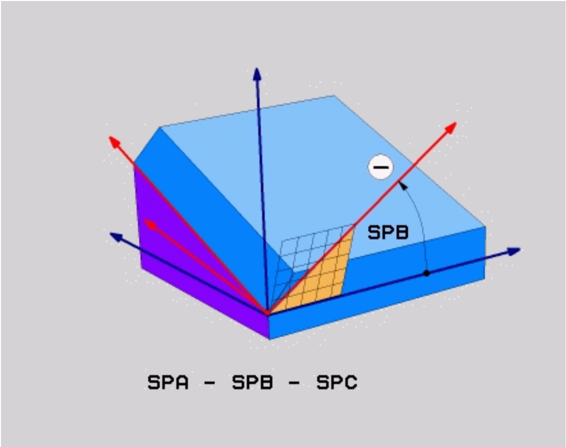
Abbreviations used

Abbreviation	Meaning
POINTS	

Defining the machining plane with a single, incremental space angle: PLANE RELATIVE

Application

Use an incremental spatial angle when an already active tilted working plane is to be tilted by **another rotation.** Example: machining a 45° chamfer on a tilted plane.



Before programming, note the following

The defined angle is always in effect in respect to the active working plane, regardless of the function you have used to activate it.

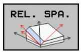
You can program any number of **`PLANE RELATIVE`** functions in a row.

If you want to return to the working plane that was active before the **`PLANE RELATIVE`** function, define the **`PLANE RELATIVE`** function again with the same angle but with the opposite algebraic sign.

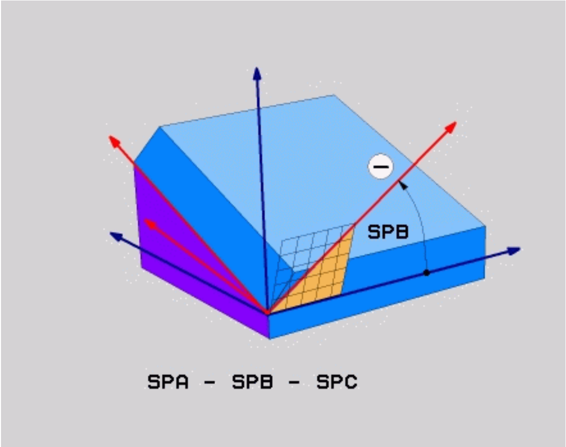
If you use the **`PLANE RELATIVE`** function in a non-tilted working plane, then you simply rotate the non-tilted plane about the spatial angle defined in the **`PLANE`** function.

Parameter description for the positioning behavior: See "Specifying the positioning behavior of the PLANE function" on page 316.

Input parameters



- **`Incremental angle?:`** Space angle about which the active machining plane is to be rotated additionally (see figure at right). Use a soft key to select the axis to be rotated about. Input range: –359.9999° to +359.9999°
- Continue with the positioning properties (see "Specifying the positioning behavior of the PLANE function" on page 316)



Abbreviations used

Abbreviation	Meaning
`RELATIVE`	

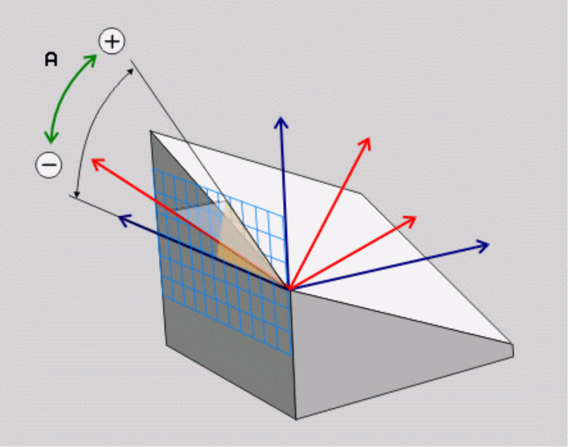
Example: NC block

```
5 PLANE RELATIVE SPB-45 .....
```

Tilting the working plane through axis angle: PLANE AXIAL (FCL 3 function)

Application

The **PLANE AXIAL** function defines both the position of the working plane and the nominal coordinates of the rotary axes. This function is particularly easy to use on machines with Cartesian coordinates and with kinematics structures in which only one rotary axis is active.



PLANE AXIAL can also be used if you have only one rotary axis active on your machine.

You can use the **PLANE RELATIVE** function after **PLANE AXIAL** if your machine allows spatial angle definitions. The machine tool manual provides further information.

Before programming, note the following

Enter only axis angles that actually exist on your machine. Otherwise the TNC generates an error message.

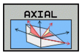
Rotary axis coordinates defined with **PLANE AXIAL** are modally effective. Successive definitions therefore build on each other. Incremental input is allowed.

Use **PLANE RESET** to reset the **PLANE AXIAL** function. Resetting by entering 0 does not deactivate **PLANE AXIAL.**

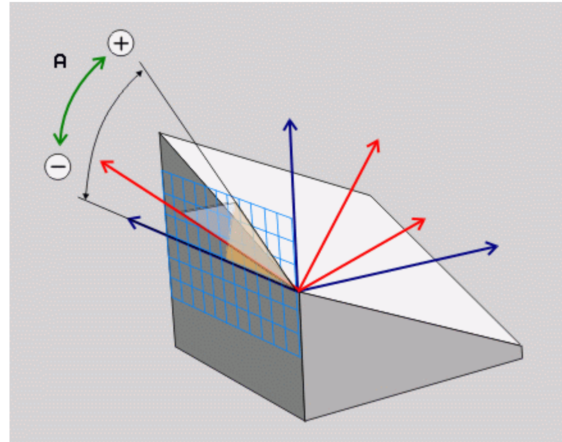
SEQ, TABLE ROT and **COORD ROT** have no function in conjunction with **PLANE AXIAL.**

Parameter description for the positioning behavior: See "Specifying the positioning behavior of the PLANE function" on page 316.

Input parameters



- **`Axis angle A?`**: Axis angle **to which** the A axis is to be tilted. If entered incrementally, it is the angle **by which** the A axis is to be tilted from its current position. Input range: –99999.9999° to +99999.9999°
- **`Axis angle B?`**: Axis angle **to which** the B axis is to be tilted. If entered incrementally, it is the angle **by which** the B axis is to be tilted from its current position. Input range: –99999.9999° to +99999.9999°
- **`Axis angle C?`**: Axis angle **to which** the C axis is to be tilted. If entered incrementally, it is the angle **by which** the C axis is to be tilted from its current position. Input range: –99999.9999° to +99999.9999°
- Continue with the positioning properties (see "Specifying the positioning behavior of the PLANE function" on page 316)



Abbreviations used

Abbreviation	Meaning
`AXIAL`	

Example: NC block

```
5 PLANE AXIAL B-45 .....
```

Specifying the positioning behavior of the PLANE function

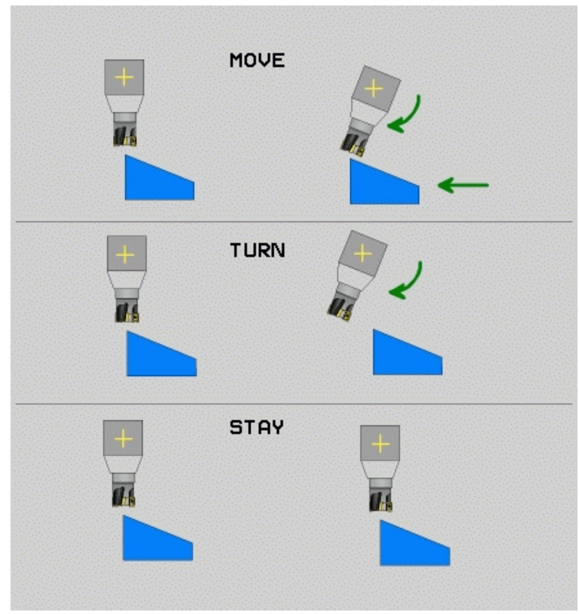
Overview

Independently of which PLANE function you use to define the tilted machining plane, the following functions are always available for the positioning behavior:

- Automatic positioning
- Selection of alternate tilting possibilities
- Selection of the type of transformation

Automatic positioning: `MOVE/TURN/STAY` (entry is mandatory)

After you have entered all parameters for the plane definition, you must specify how the rotary axes will be positioned to the calculated axis values:



MOVE

- The PLANE function is to automatically position the rotary axes to the calculated position values. The position of the tool relative to the workpiece is to remain the same. The TNC carries out a compensation movement in the linear axes.

TURN

- The PLANE function is to automatically position the rotary axes to the calculated position values, but only the rotary axes are positioned. The TNC does **not** carry out a compensation movement in the linear axes

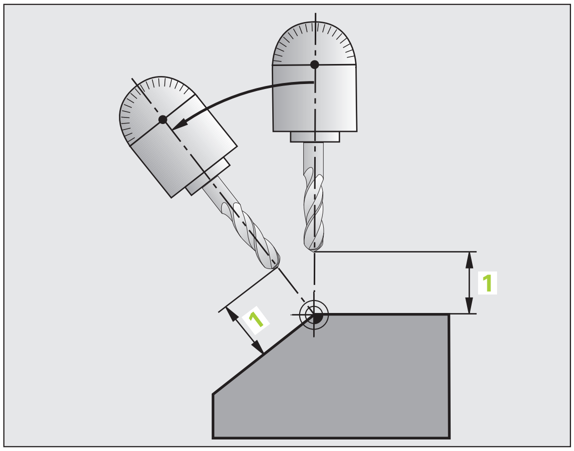
STAY

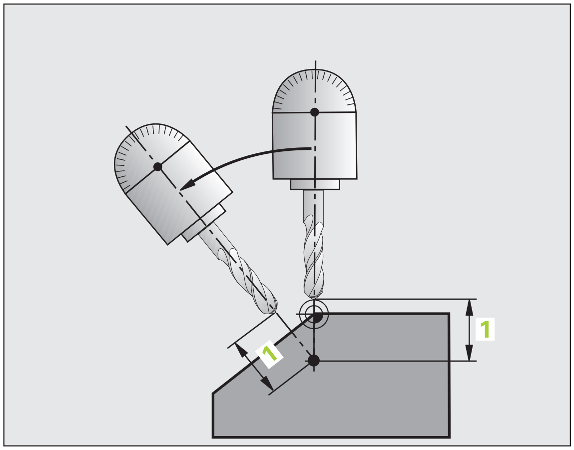
- You will position the rotary axes later in a separate positioning block

If you have selected the **MOVE** option (**PLANE** function is to position the axes automatically), the following two parameters must still be defined: **Dist. tool tip – center of rot.** and **Feed rate? F=**. If you have selected the **TURN** option (**PLANE** function is to position the axes automatically without any compensating movement), the following parameter must still be defined: **Feed rate? F=**. As an alternative to defining a feed rate **F** directly by numerical value, you can also position with **FMAX** (rapid traverse) or **FAUTO** (feed rate from the **TOOL CALLT** block).

> If you use **PLANE AXIAL** together with **STAY,** you have to position the rotary axes in a separated block after the **PLANE** function.

▶ **Dist. tool tip – center of rot.** (incremental): The TNC tilts the tool (or table) relative to the tool tip. The **DIST** parameter shifts the center of rotation of the positioning movement relative to the current position of the tool tip.





Note:

- If the tool is already at the given distance to the workpiece before positioning, then relatively speaking the tool is at the same position after positioning (see figure at center right, 1 = DIST)
- If the tool is not at the given distance to the workpiece before positioning, then relatively speaking the tool is offset from the original position after positioning (see figure at bottom right, 1= DIST)

▶ **Feed rate? F=**: Contour speed at which the tool should be positioned

Positioning the rotary axes in a separate block

Proceed as follows if you want to position the rotary axes in a separate positioning block (option **STAY** selected):

Pre-position the tool to a position where there is no danger of collision with the workpiece (clamping devices) during positioning.

▶ Select any **PLANE** function, and define automatic positioning with the **STAY** option. During program execution the TNC calculates the position values of the rotary axes present on the machine, and stores them in the system parameters Q120 (A axis), Q121 (B axis) and Q122 (C axis)

▶ Define the positioning block with the angular values calculated by the TNC

NC example blocks: Position a machine with a rotary table C and a tilting table A to a spatial angle of B+45°.

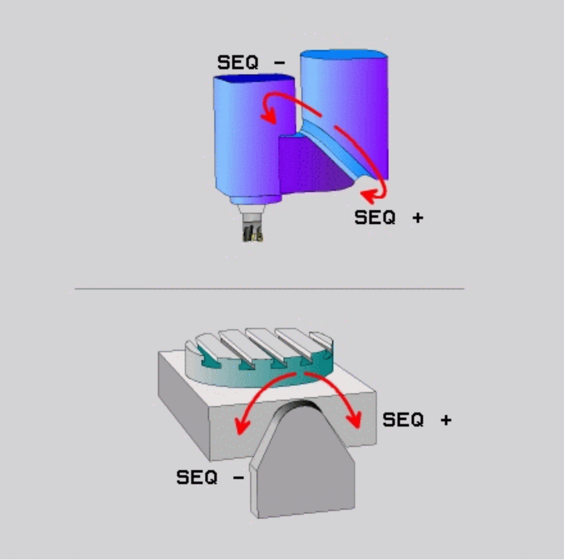
...	
12 L Z+250 R0 FMAX	Position at clearance height.
13 PLANE SPATIAL SPA+0 SPB+45 SPC+0 STAY	Define and activate the PLANE function
14 L A+Q120 C+Q122 F2000	Position the rotary axis with the values calculated by the TNC
...	Define machining in the tilted working plane

Selection of alternate tilting possibilities: SEQ +/– (entry optional)

The position you define for the working plane is used by the TNC to calculate the appropriate positioning of the rotary axes present on the machine. In general there are always two solution possibilities.

Use the **SEQ** switch to specify which possibility the TNC should use:

- **SEQ+** positions the master axis so that it assumes a positive angle. The master axis is the 1st rotary axis from the tool, or the last rotary axis from the table (depending on the machine configuration (see figure at top right)).
- **SEQ-** positions the master axis so that it assumes a negative angle.



If the solution you chose with **SEQ** is not within the machine's range of traverse, the TNC displays the **Entered angle not permitted** error message.

> When the **PLANE AXIS** function is used, the **SEQ** switch is nonfunctional.

If you do not define **SEQ,** the TNC determines the solution as follows:

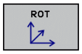
1. The TNC first checks whether both solution possibilities are within the traverse range of the rotary axes
2. If they are, then the TNC selects the shortest possible solution
3. If only one solution is within the traverse range, the TNC selects this solution
4. If neither solution is within the traverse range, the TNC displays the **Entered angle not permitted** error message

Example for a machine with a rotary table C and a tilting table A.
Programmed function: **PLANE SPATIAL SPA+0 SPB+45 SPC+0**

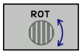
Limit switch	Starting position	SEQ	Resulting axis position
None	A+0, C+0	not prog.	A+45, C+90
None	A+0, C+0	+	A+45, C+90
None	A+0, C+0	–	A–45, C–90
None	A+0, C–105	not prog.	A–45, C–90
None	A+0, C–105	+	A+45, C+90
None	A+0, C–105	–	A–45, C–90
–90 < A < +10	A+0, C+0	not prog.	A–45, C–90
–90 < A < +10	A+0, C+0	+	Error message
None	A+0, C–135	+	A+45, C+90

Selecting the type of transformation (entry optional)

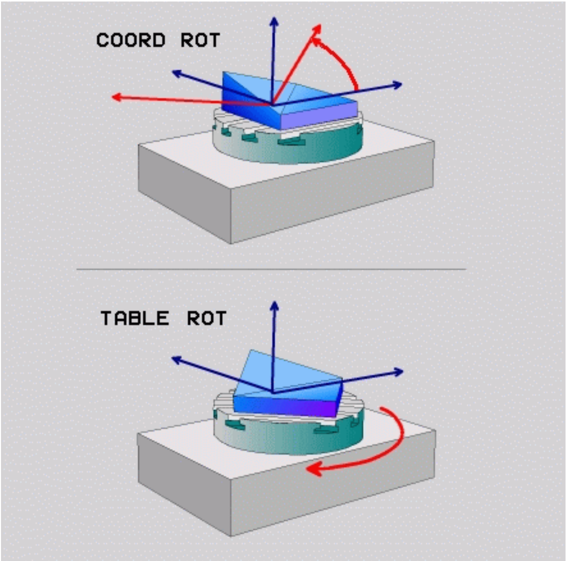
On machines with C-rotary tables, a function is available for specifying the type of transformation:



- **COORD ROT** specifies that the PLANE function should only rotate the coordinate system to the defined tilting angle. The rotary table is not moved; the compensation is purely mathematical.



- **TABLE ROT** specifies that the PLANE function should position the rotary table to the defined tilting angle. Compensation results from rotating the workpiece.



When the **PLANE AXIAL** function is used, **COORD ROT** and **TABLE ROT** are nonfunctional.

If you use the **TABLE ROT** function in conjunction with a basic rotation and a tilting angle of 0, then the TNC tilts the table to the angle defined in the basic rotation.

11.3 Miscellaneous Functions for Rotary Axes

Feed rate in mm/min on rotary axes A, B, C: M116 (software option 1)

Standard behavior

The TNC interprets the programmed feed rate of a rotary axis in degrees/min (in mm programs and also in inch programs). The feed rate therefore depends on the distance from the tool center to the center of axis rotation.

The larger this distance becomes, the greater the contouring feed rate.

Feed rate in mm/min on rotary axes with M116

The machine geometry must be specified by the machine tool builder in the description of kinematics.

M116 works only on rotary tables. M116 cannot be used with swivel heads. If your machine is equipped with a table/head combination, the TNC ignores the swivel-head rotary axes.

M116 is also effective in an active tilted working plane.

The TNC interprets the programmed feed rate of a rotary axis in degrees/min (or 1/10 inch/min). In this case, the TNC calculates the feed for the block at the start of each block. With a rotary axis, the feed rate is not changed during execution of the block even if the tool moves toward the center of the rotary axis.

Effect

M116 is effective in the working plane. With M117 you can reset M116. M116 is also canceled at the end of the program.

M116 becomes effective at the start of block.

Shorter-path traverse of rotary axes: M126

Standard behavior

The standard behavior of the TNC while positioning rotary axes whose display has been reduced to values less than 360° is dependent on machine parameter **`shortestDistance`** (300401). This machine parameter defines whether the TNC should consider the difference between nominal and actual position, or whether it should always (even without M126) choose the shortest path to the programmed position. Examples:

Actual position	Nominal position	Traverse
350°	10°	–340°
10°	340°	+330°

Behavior with M126

With M126, the TNC will move the axis on the shorter path of traverse for rotary axes whose display is reduced to values less than 360°. Examples:

Actual position	Nominal position	Traverse
350°	10°	+20°
10°	340°	–30°

Effect

M126 becomes effective at the start of block.

To cancel M126, enter M127. At the end of program, M126 is automatically canceled.

Reducing display of a rotary axis to a value less than 360°: M94

Standard behavior

The TNC moves the tool from the current angular value to the programmed angular value.

Example:

Current angular value:	538°
Programmed angular value:	180°
Actual distance of traverse:	-358°

Behavior with M94

At the start of block, the TNC first reduces the current angular value to a value less than 360° and then moves the tool to the programmed value. If several rotary axes are active, M94 will reduce the display of all rotary axes. As an alternative you can enter a rotary axis after M94. The TNC then reduces the display only of this axis.

Example NC blocks

To reduce display of all active rotary axes:

```
N50 M94 *
```

To reduce display of the C axis only:

```
N50 M94 C *
```

To reduce display of all active rotary axes and then move the tool in the C axis to the programmed value:

```
N50 G00 C+180 M94 *
```

Effect

M94 is effective only in the block in which it is programmed.

M94 becomes effective at the start of block.

Selecting tilting axes: M138

Standard behavior

The TNC performs M128 and TCPM, and tilts the working plane, only in those axes for which the machine tool builder has set the appropriate machine parameters.

Behavior with M138

The TNC performs the above functions only in those tilting axes that you have defined using M138.

Effect

M138 becomes effective at the start of block.

You can reset M138 by reprogramming it without entering any axes.

Example NC blocks

Perform the above-mentioned functions only in the tilting axis C:

```
N50 G00 Z+100 R0 M138 C *
```

Compensating the machine's kinematics configuration for ACTUAL/NOMINAL positions at end of block: M144 (software option 2)

Standard behavior

The TNC moves the tool to the positions given in the part program. If the position of a tilted axis changes in the program, the resulting offset in the linear axes must be calculated, and traversed in a positioning block.

Behavior with M144

The TNC calculates into the position value any changes in the machine's kinematics configuration which result, for example, from adding a spindle attachment. If the position of a controlled tilted axis changes, the position of the tool tip to the workpiece is also changed. The resulting offset is calculated in the position display.

Positioning blocks with M91/M92 are permitted if M144 is active.

The position display in the operating modes FULL SEQUENCE and SINGLE BLOCK does not change until the tilting axes have reached their final position.

Effect

M144 becomes effective at the start of the block. M144 does not function in connection with M128 or a tilted working plane.

You can cancel M144 by programming M145.

The machine geometry must be specified by the machine tool builder in the description of kinematics.

The machine tool builder determines the behavior in the automatic and manual operating modes. Refer to your machine manual.

12

Manual Operation and Setup

12.1 Switch-On, Switch-Off

Switch-on

Switch-on and crossing over the reference points can vary depending on the machine tool. Refer to your machine manual.

Switch on the power supply for control and machine. The TNC then displays the following dialog:

SYSTEM STARTUP

TNC is started

POWER INTERRUPTED

CE — TNC message that the power was interrupted—clear the message

COMPILE A PLC PROGRAM

The PLC program of the TNC is compiled automatically.

RELAY EXT. DC VOLTAGE MISSING

I — Switch on external dc voltage The TNC checks the functioning of the EMERGENCY STOP circuit.

MANUAL OPERATION
TRAVERSE REFERENCE POINTS

I — Cross the reference points manually in the displayed sequence: For each axis press the machine START button, or

X Y — Cross the reference points in any sequence: Press and hold the machine axis direction button for each axis until the reference point has been traversed.

If your machine is equipped with absolute encoders, you can leave out crossing the reference marks. In such a case, the TNC is ready for operation immediately after the machine control voltage is switched on.

The TNC is now ready for operation in the Manual Operation mode.

> The reference points need only be crossed if the machine axes are to be moved. If you intend only to write, edit or test programs, you can select the Programming and Editing or Test Run modes of operation immediately after switching on the control voltage.
>
> You can cross the reference points later by pressing the PASS OVER REFERENCE MARK soft key in the Manual Operation mode.

Crossing the reference point in a tilted working plane

Danger of collision!

Make sure that the angle values entered in the menu for tilting the working plane match the actual angles of the tilted axis.

Deactivate the "Tilt Working Plane" function before you cross the reference points. Take care that there is no collision. Retract the tool from the current position first, if necessary.

The TNC automatically activates the tilted working plane if this function was enabled when the control was switched off. Then the TNC moves the axes in the tilted coordinate system when an axis-direction key is pressed. Position the tool in such a way that a collision is excluded during the subsequent crossing of the reference points. To cross the reference points you have to deactivate the "Tilt Working Plane" function, see "Activating manual tilting", page 360.

If you use this function, then for non-absolute encoders you must confirm the positions of the rotary axes, which the TNC displays in a pop-up window. The position displayed is the last active position of the rotary axes before switch-off.

If one of the two functions that were active before is active now, the NC START button has no function. The TNC outputs a corresponding error message.

Switch-off

To prevent data from being lost at switch-off, you need to shut down the operating system of the TNC as follows:

- Select the Manual Operation mode

OFF

- Select the function for shutting down, confirm again with the YES soft key
- When the TNC displays the message **NOW IT IS SAFE TO TURN POWER OFF** in a superimposed window, you may cut off the power supply to the TNC

Inappropriate switch-off of the TNC can lead to data loss!

Remember that pressing the END key after the control has been shut down restarts the control. Switch-off during a restart can also result in data loss!

12.2 Moving the Machine Axes

Note

Traversing with the machine axis direction buttons can vary depending on the machine tool. The machine tool manual provides further information.

Moving the axis using the machine axis direction buttons

Select the Manual Operation mode

X — Press the machine axis direction button and hold it as long as you wish the axis to move, or

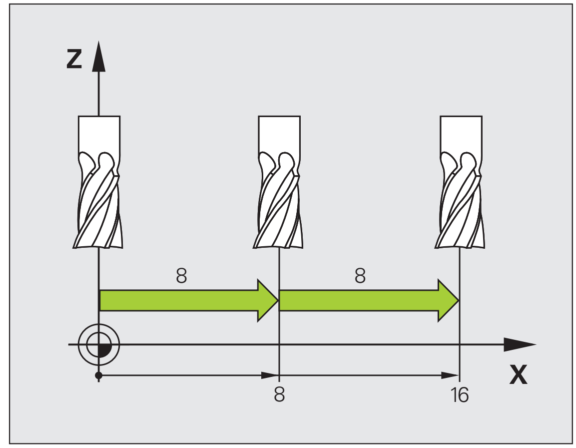
X I — Move the axis continuously: Press and hold the machine axis direction button, then press the machine START button

0 — To stop the axis, press the machine STOP button

You can move several axes at a time with these two methods. You can change the feed rate at which the axes are traversed with the F soft key, see "Spindle Speed S, Feed Rate F and Miscellaneous Functions M", page 332.

Incremental jog positioning

With incremental jog positioning you can move a machine axis by a preset distance.



Select the Manual Operation or Electronic Handwheel mode

Shift the soft-key row

INCRE-MENT OFF ON

Select incremental jog positioning: Switch the INCREMENT soft key to ON

JOG INCREMENT =

ENT

Enter the jog increment in mm, and confirm with the ENT key

X

Press the machine axis direction button as often as desired

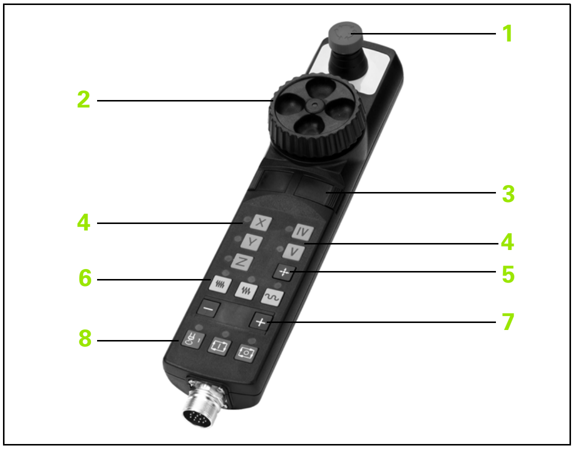
The maximum permissible value for infeed is 10 mm.

Traversing with the HR 410 electronic handwheel

The portable HR 410 handwheel is equipped with two permissive buttons. The permissive buttons are located below the star grip.

You can only move the machine axes when a permissive button is depressed (machine-dependent function).

The HR 410 handwheel features the following operating elements:



1 EMERGENCY STOP button
2 Handwheel
3 Permissive buttons/keys
4 Axis address keys
5 Actual-position-capture key
6 Keys for defining the feed rate (slow, medium, fast; the feed rates are set by the machine tool builder)
7 Direction in which the TNC moves the selected axis
8 Machine function (set by the machine tool builder)

The red indicator lights show the axis and feed rate you have selected.

It is also possible to move the machine axes with the handwheel during program run if **M118** is active.

Procedure

Select the Electronic Handwheel operating mode

Press and hold a permissive button

Select the axis

Select the feed rate

Move the active axis in the positive direction, or

Move the active axis in the negative direction

12.3 Spindle Speed S, Feed Rate F and Miscellaneous Functions M

Function

In the Manual Operation and Electronic Handwheel operating modes, you can enter the spindle speed S, feed rate F and the miscellaneous functions M with soft keys. The miscellaneous functions are described in Chapter 7 "Programming: Miscellaneous Functions."

The machine tool builder determines which miscellaneous functions M are available on your control and what effects they have.

Entering values

Spindle speed S, miscellaneous function M

S — To enter the spindle speed, press the S soft key.

SPINDLE SPEED S =

1000 I — Enter the desired spindle speed and confirm your entry with the machine START button.

The spindle speed S with the entered rpm is started with a miscellaneous function M. Proceed in the same way to enter a miscellaneous function M.

Feed rate F

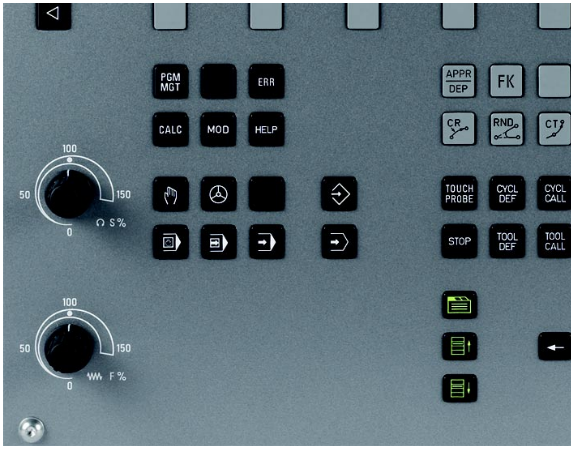
After entering a feed rate F, you must confirm your entry with the ENT key instead of the machine START button.

The following is valid for feed rate F:

- If you enter F=0, then the lowest feed rate from the machine parameter **manualFeed** is effective.
- If the feed rate entered exceeds the value defined in the machine parameter **maxFeed,** then the parameter value is effective.
- F is not lost during a power interruption

Changing the spindle speed and feed rate

With the override knobs you can vary the spindle speed S and feed rate F from 0% to 150% of the set value.



> The override knob for spindle speed is only functional on machines with infinitely variable spindle drive.

12.4 Datum Setting without a 3-D Touch Probe

Note

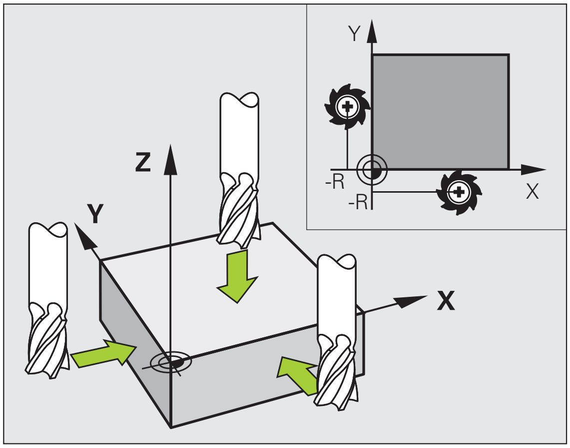
Datum setting with a 3-D touch probe: (see "Set the datum with a 3-D touch probe" on page 350).

You fix a datum by setting the TNC position display to the coordinates of a known position on the workpiece.

Preparation

- Clamp and align the workpiece
- Insert the zero tool with known radius into the spindle
- Ensure that the TNC is showing the actual position values

Workpiece presetting with axis keys



Protective measure

If the workpiece surface must not be scratched, you can lay a metal shim of known thickness d on it. Then enter a tool axis datum value that is larger than the desired datum by the value d.

Select the **Manual Operation** mode.

X Y Z Move the tool slowly until it touches (scratches) the workpiece surface

Z Select the axis

DATUM SETTING Z=

0 ENT Zero tool in spindle axis: Set the display to a known workpiece position (here, 0) or enter the thickness d of the shim. In the tool axis, offset the tool radius

Repeat the process for the remaining axes.

If you are using a preset tool, set the display of the tool axis to the length L of the tool or enter the sum Z=L+d

The TNC automatically saves the datum set with the axis keys in line 0 of the preset table.

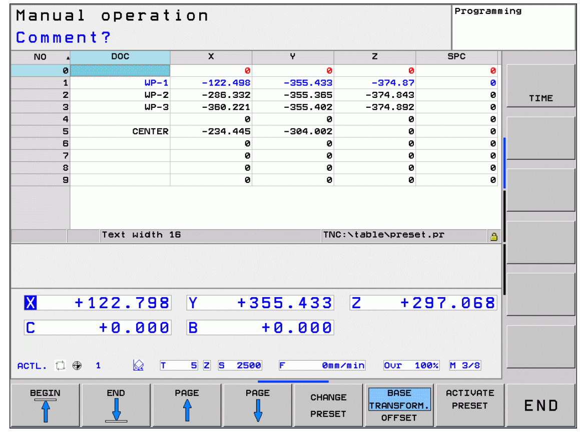
Datum management with the preset table

You should definitely use the preset table if:

- Your machine is equipped with rotary axes (tilting table or swivel head) and you work with the function for tilting the working plane
- Your machine is equipped with a spindle-head changing system
- Up to now you have been working with older TNC controls with REF-based datum tables
- You wish to machine identical workpieces that are differently aligned

The preset table can contain any number of lines (datums). To optimize the file size and the processing speed, you should use only as many lines as you need for datum management.

For safety reasons, new lines can be inserted only at the end of the preset table.



Saving the datums in the preset table

The preset table has the name **PRESET.PR,** and is saved in the directory **TNC:\table. PRESET.PR** is editable in the **Manual** and **El. Handwheel** modes only if the **CHANGE PRESET** soft key was pressed.

It is permitted to copy the preset table into another directory (for data backup). Lines that were written by your machine tool builder are also always write-protected in the copied tables. You therefore cannot edit them.

Never change the number of lines in the copied tables! That could cause problems when you want to reactivate the table.

To activate the preset table copied to another directory you have to copy it back to the directory **TNC:\table**.

There are several methods for saving datums and/or basic rotations in the preset table:

- Through probing cycles in the **Manual Operation** or **Electronic Handwheel** modes (see Chapter 14)
- Through the probing cycles 400 to 402 and 410 to 419 in automatic mode (see User's Manual, Cycles, Chapters 14 and 15)
- Manual entry (see description below)

Basic rotations from the preset table rotate the coordinate system about the preset, which is shown in the same line as the basic rotation.

Remember to ensure that the position of the tilting axes matches the corresponding values of the 3-D ROT menu when setting the datum. Therefore:

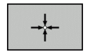
- If the "Tilt working plane" function is not active, the position display for the rotary axes must be = 0° (zero the rotary axes if necessary).
- If the "Tilt working plane" function is active, the position displays for the rotary axes must match the angles entered in the 3-D ROT menu.

The line 0 in the preset table is write protected. In line 0, the TNC always saves the datum that you most recently set manually via the axis keys or via soft key. If the datum set manually is active, the TNC displays the text **PR MAN(0)** in the status display.

Manually saving the datums in the preset table

In order to set datums in the preset table, proceed as follows:

	Select the **Manual Operation** mode.
X Y Z	Move the tool slowly until it touches (scratches) the workpiece surface, or position the measuring dial correspondingly
PRESET TABLE	Displaying the preset table: The TNC opens the preset table and sets the cursor to the active table row
CHANGE PRESET	Select functions for entering the presets: The TNC displays the available possibilities for entry in the soft-key row. See the table below for a description of the entry possibilities
	Select the line in the preset table that you want to change (the line number is the preset number)
	If needed, select the column (axis) in the preset table that you want to change
CORRECT THE PRESET	Use the soft keys to select one of the available entry possibilities (see the following table)

Function	Soft key
Directly transfer the actual position of the tool (the measuring dial) as the new datum: This function only saves the datum in the axis which is currently highlighted.	
Assign any value to the actual position of the tool (the measuring dial): This function only saves the datum in the axis which is currently highlighted. Enter the desired value in the pop-up window.	ENTER NEW PRESET
Incrementally shift a datum already stored in the table: This function only saves the datum in the axis which is currently highlighted. Enter the desired corrective value with the correct sign in the pop-up window. If inch display is active: enter the value in inches, and the TNC will internally convert the entered values to mm.	CORRECT THE PRESET
Directly enter the new datum without calculation of the kinematics (axis-specific). Only use this function if your machine has a rotary table, and you want to set the datum to the center of the rotary table by entering 0. This function only saves the datum in the axis which is currently highlighted. Enter the desired value in the pop-up window. If inch display is active: enter the value in inches, and the TNC will internally convert the entered values to mm.	EDIT CURRENT FIELD
Select the BASIC TRANSFORMATION/AXIS OFFSET view. The BASIC TRANSFORMATION view shows the X, Y and Z columns. Depending on the machine, the SPA, SPB and SPC columns are displayed additionally. Here, the TNC saves the basic rotation (for the Z tool axis, the TNC uses the SPC column). The OFFSET view shows the offset values to the preset.	BASE TRANSFORM. OFFSET
Write the currently active datum to a selectable line in the table: This function saves the datum in all axes, and then activates the appropriate row in the table automatically. If inch display is active: enter the value in inches, and the TNC will internally convert the entered values to mm.	SAVE PRESET

Editing the preset table

Editing function in table mode	Soft key
Select beginning of table	BEGIN
Select end of table	END
Select previous page in table	PAGE
Select next page in table	PAGE
Select the functions for preset entry	CHANGE PRESET
Display Basic Transformation/Axis Offset selection	BASE TRANSFORM. OFFSET
Activate the datum of the selected line of the preset table	ACTIVATE PRESET
Add the entered number of lines to the end of the table (2nd soft-key row)	APPEND N LINES
Copy the highlighted field (2nd soft-key row)	COPY FIELD
Insert the copied field (2nd soft-key row)	PASTE FIELD
Reset the selected line: The TNC enters—in all columns (2nd soft-key row)	RESET LINE
Insert a single line at the end of the table (2nd soft-key row)	INSERT LINE
Delete a single line at the end of the table (2nd soft-key row)	DELETE LINE

Activating a datum from the preset table in the Manual Operation mode

When activating a datum from the preset table, the TNC resets the active datum shift, mirroring, rotation and scaling factor.

However, a coordinate transformation that was programmed in Cycle 19 Tilted Working Plane, or through the PLANE function, remains active.

- Select the **Manual Operation** mode.
- PRESET TABLE: Display the preset table
- Select the datum number you want to activate, or
- GOTO 4 ENT: With the GOTO key, select the datum number that you want to activate. Confirm with the ENT key
- ACTIVATE PRESET: Activate the preset
- EXECUTE: Confirm activation of the datum. The TNC sets the display and—if defined—the basic rotation
- END: Exit the preset table

Activating the datum from the preset table in an NC program

To activate datums from the preset table during program run, use Cycle 247. In Cycle 247 you define the number of the datum that you want to activate (see User's Manual, Cycles, Cycle 247 SET DATUM).

12.5 Using the 3-D Touch Probe

Overview

The following touch probe cycles are available in the Manual Operation mode:

> HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.
>
> If you use probing functions in a tilted working plane, you must set 3-D ROT to **`Active`** for the Manual Operation and Automatic operating modes.

Function	Soft key	Page
Calibrating the effective length	CAL L	Page 345
Calibrating the effective radius	CAL R	Page 346
Measuring a basic rotation using a line	ROTATION	Page 349
Setting the datum in any axis	PROBING POS	Page 350
Setting a corner as datum	PROBING P	Page 351
Setting a circle center as datum	PROBING CC	Page 352
Touch probe system data management	TCH PROBE TABLE	See User's Manual for Cycles

> For more information about the touch probe table, refer to the User's Manual for Cycle Programming.

Selecting probe cycles

- To select the Manual Operation or El. Handwheel mode of operation

TOUCH PROBE

- Select the touch probe functions by pressing the TOUCH PROBE soft key. The TNC displays additional soft keys: see table above.

ROTATION

- To select the probe cycle, press the appropriate soft key, for example PROBING ROT, and the TNC displays the associated menu.

Writing the measured values from touch probe cycles in datum tables

Use this function if you want to save measured values in the workpiece coordinate system. If you want to save measured values in the fixed machine coordinate system (REF coordinates), press the ENTER IN PRESET TABLE soft key (see "Writing the measured values from touch probe cycles in the preset table" on page 343).

With the ENTER IN DATUM TABLE soft key, the TNC can write the values measured during a touch probe cycle in a datum table:

- Select any probe function
- Enter the desired coordinates of the datum in the appropriate input boxes (depends on the touch probe cycle being run)
- Enter the datum number in the **`Number in table=`** input box
- Press the ENTER IN DATUM TABLE soft key. The TNC saves the datum in the indicated datum table under the entered number

Writing the measured values from touch probe cycles in the preset table

Use this function if you want to save measured values in the machine-based coordinate system (REF coordinates). If you want to save measured values in the workpiece coordinate system, press the ENTER IN DATUM TABLE soft key (see "Writing the measured values from touch probe cycles in datum tables" on page 343).

With the ENTER IN PRESET TABLE soft key, the TNC can write the values measured during a probe cycle in the preset table. The measured values are then stored referenced to the machine-based coordinate system (REF coordinates). The preset table has the name PRESET.PR, and is saved in the directory TNC:\table\.

- Select any probe function
- Enter the desired coordinates of the datum in the appropriate input boxes (depends on the touch probe cycle being run)
- Enter the preset number in the **`Number in table:`** input box
- Press the ENTER IN PRESET TABLE soft key. The TNC saves the datum in the preset table under the entered number

12.6 Calibrating the 3-D Touch Probe

Introduction

In order to precisely specify the actual trigger point of a 3-D touch probe, you must calibrate the touch probe, otherwise the TNC cannot provide precise measuring results.

Always calibrate a touch probe in the following cases:

- Commissioning
- Stylus breakage
- Stylus exchange
- Change in the probe feed rate
- Irregularities caused, for example, when the machine heats up
- Change of active tool axis

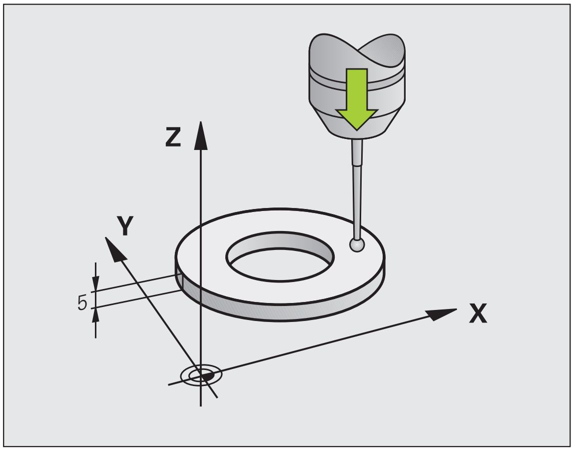
During calibration, the TNC finds the "effective" length of the stylus and the "effective" radius of the ball tip. To calibrate the 3-D touch probe, clamp a ring gauge of known height and known internal radius to the machine table.

Calibrating the effective length

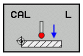
HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

If you use probing functions in a tilted working plane, you must set 3-D ROT to **Active** for the Manual and Automatic operating modes.

The effective length of the touch probe is always referenced to the tool datum. The machine tool builder usually defines the spindle tip as the tool datum.



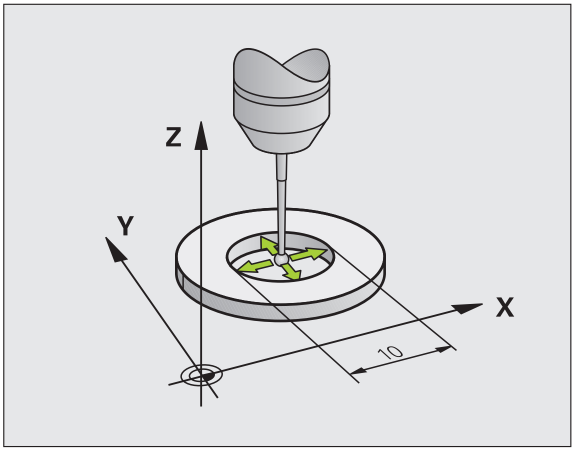
- Set the datum in the spindle axis such that for the machine tool table Z=0.



- To select the calibration function for the touch probe length, press the TOUCH PROBE and CAL. L soft keys. The TNC then displays a menu window with four input fields.
- Enter the tool axis (with the axis key)
- **Datum**: Enter the height of the ring gauge
- **Effective ball radius** and **Effective length** do not require input
- Move the touch probe to a position just above the ring gauge
- To change the traverse direction (if necessary), press a soft key or an arrow key
- To probe the upper surface of the ring gauge, press the machine START button

Calibrating the effective radius and compensating center misalignment

> HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.
>
> If you use probing functions in a tilted working plane, you must set 3-D ROT to **`Active`** for the Manual Operation and Automatic operating modes.



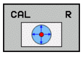
After the touch probe is inserted, it normally needs to be aligned exactly with the spindle axis. The calibration function determines the misalignment between touch probe axis and spindle axis and computes the compensation.

The calibration routine varies depending on the entry in the TRACK column of the touch probe table (spindle orientation active/inactive). If the function for orienting the infrared touch probe to the programmed probe direction is active, the calibration cycle is executed after you have pressed NC Start once. If the function is not active, you can decide whether you want to compensate the center misalignment by calibrating the effective radius.

The TNC rotates the 3-D touch probe by 180° for calibrating the center misalignment. The rotation is initiated by a miscellaneous function that is set by the machine tool builder in Machine Parameter mStrobeUTurn.

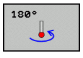
Proceed as follows for manual calibration:

- In the Manual Operation mode, position the ball tip in the bore of the ring gauge



- To select the calibration function for the ball-tip radius and the touch probe center misalignment, press the CAL. R soft key
- Select the tool axis and enter the radius of the ring gauge
- To probe the workpiece, press the machine START button four times. The 3-D touch probe contacts a position on the hole in each axis direction and calculates the effective ball-tip radius
- If you want to terminate the calibration function at this point, press the END soft key

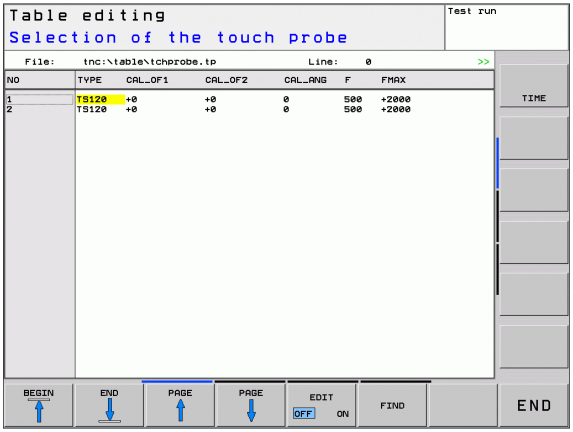
> In order to be able to determine ball-tip center misalignment, the TNC needs to be specially prepared by the machine manufacturer. The machine tool manual provides further information.



- If you want to determine the ball-tip center misalignment, press the 180° soft key. The TNC rotates the touch probe by 180°
- To probe the workpiece, press the machine START button four times. The 3-D touch probe contacts a position on the hole in each axis direction and calculates the ball-tip center misalignment

Show calibration values

The TNC saves the effective length and effective radius of the touch probe in the tool table. The TNC saves the ball-tip center misalignment in the touch-probe table, in the **CAL_OF1** (principal axis) and **CAL_OF2** (minor axis) columns. You can display the values on the screen by pressing the TOUCH-PROBE TABLE soft key.



> Make sure that you have activated the correct tool number before using the touch probe, regardless of whether you wish to run the touch probe cycle in automatic mode or manual mode.
>
> The determined calibration values are not considered until a tool is called (or called again, if required).

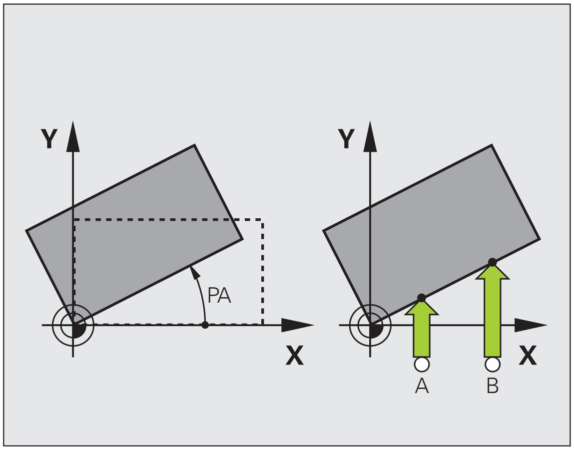
> For more information about the touch probe table, refer to the User's Manual for Cycle Programming.

12.7 Compensating Workpiece Misalignment with a 3-D Touch Probe

Introduction

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

If you use probing functions in a tilted working plane, you must set 3-D ROT to **`Active`** for the Manual Operation and Automatic operating modes.



The TNC electronically compensates workpiece misalignment by computing a "basic rotation."

For this purpose, the TNC sets the rotation angle to the desired angle with respect to the reference axis in the working plane. See figure at right.

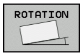
The TNC saves the basic rotation, depending on the tool axis, in the columns SPA, SPB or SPC of the preset table.

Select the probe direction perpendicular to the angle reference axis when measuring workpiece misalignment.

To ensure that the basic rotation is calculated correctly during program run, program both coordinates of the working plane in the first positioning block.

You can also use a basic rotation in conjunction with the PLANE function. In this case, first activate the basic rotation and then the PLANE function.

Measuring the basic rotation



- Select the probe function by pressing the PROBING ROT soft key
- Position the touch probe at a position near the first touch point
- Select the probe direction perpendicular to the angle reference axis: Select the axis by soft key
- To probe the workpiece, press the machine START button
- Position the touch probe at a position near the second touch point
- To probe the workpiece, press the machine START button. The TNC determines the basic rotation and displays the angle after the dialog **`Rotation angle =`**
- Activate basic rotation: Press the SET BASIC ROTATION soft key
- Terminate the probe function by pressing the END soft key

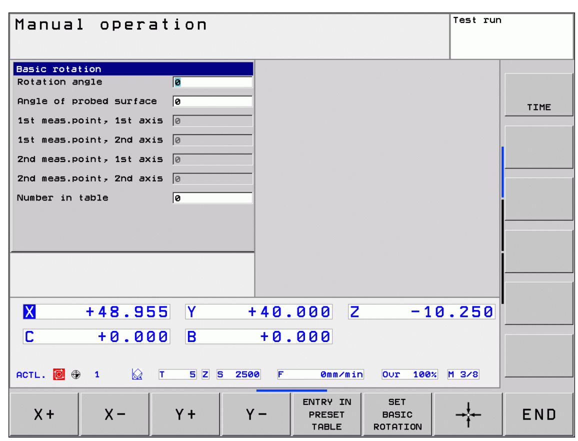
Saving the basic rotation in the preset table

- After the probing process, enter the preset number in which the TNC is to save the active basic rotation in the **`Number in table:`** input box
- Press the ENTRY IN PRESET TABLE soft key to save the basic rotation in the preset table

Displaying a basic rotation

The angle of the basic rotation appears after ROTATION ANGLE whenever PROBING ROT is selected. The TNC also displays the rotation angle in the additional status display (STATUS POS.)

In the status display a symbol is shown for a basic rotation whenever the TNC is moving the axes according to a basic rotation.



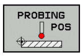
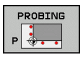
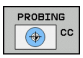
Canceling a basic rotation

- Select the probe function by pressing the PROBING ROT soft key
- Enter a rotation angle of zero and confirm with the SET BASIC ROTATION soft key
- Terminate the probe function by pressing the END soft key

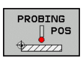
12.8 Set the datum with a 3-D touch probe

Overview

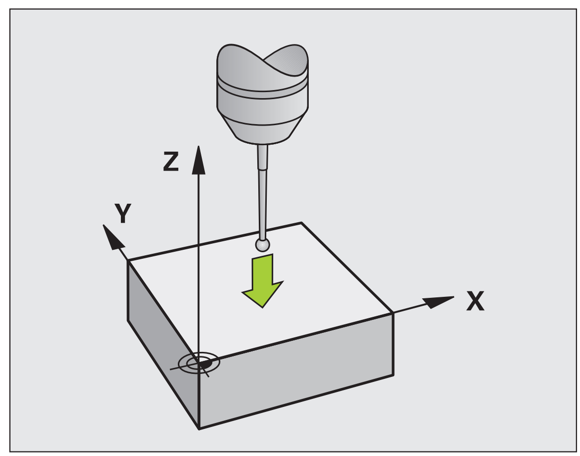
The following soft-key functions are available for setting the datum on an aligned workpiece:

Soft key	Function	Page
	Datum setting in any axis	Page 350
	Setting a corner as datum	Page 351
	Setting a circle center as datum	Page 352

Datum setting in any axis



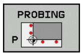
- Select the probe function by pressing the PROBING POS soft key
- Move the touch probe to a position near the touch point
- Use the soft keys to select the probe axis and direction in which you want to set the datum, such as Z in direction Z–
- To probe the workpiece, press the machine START button
- **Datum:** Enter the nominal coordinate and confirm your entry with the SET DATUM soft key, see "Writing the measured values from touch probe cycles in datum tables", page 343
- To terminate the probe function, press the END soft key

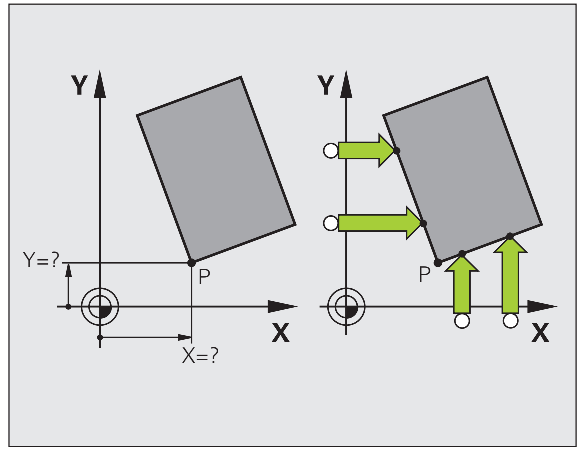


HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

If you use probing functions in a tilted working plane, you must set 3-D ROT to **Active** for the Manual Operation and Automatic operating modes.

Corner as datum





- Select the probe function by pressing the PROBING P soft key.
- Position the touch probe near the first touch point on the first workpiece edge
- Select the probe direction by soft key
- To probe the workpiece, press the machine START button
- Position the touch probe near the second touch point on the same workpiece edge
- To probe the workpiece, press the machine START button
- Position the touch probe near the first touch point on the second workpiece edge
- Select the probe direction by soft key
- To probe the workpiece, press the machine START button
- Position the touch probe near the second touch point on the same workpiece edge
- To probe the workpiece, press the machine START button
- **Datum:** Enter both datum coordinates into the menu window, and confirm your entry with the SET DATUM soft key, or see "Writing the measured values from touch probe cycles in the preset table", page 343
- To terminate the probe function, press the END soft key

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

If you use probing functions in a tilted working plane, you must set 3-D ROT to **Active** for the Manual Operation and Automatic operating modes.

Circle center as datum

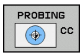
With this function, you can set the datum at the center of bore holes, circular pockets, cylinders, studs, circular islands, etc.

Inside circle:

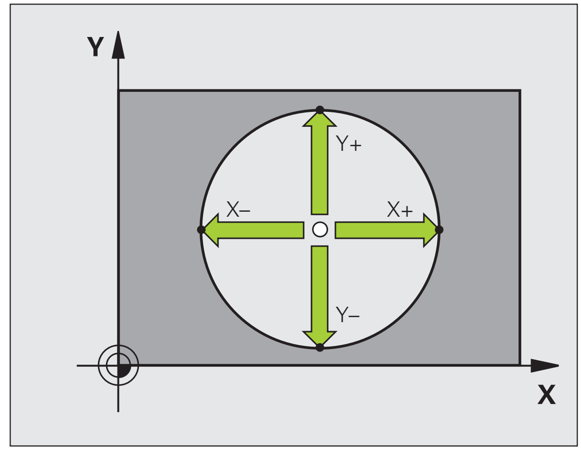
The TNC automatically probes the inside wall in all four coordinate axis directions.

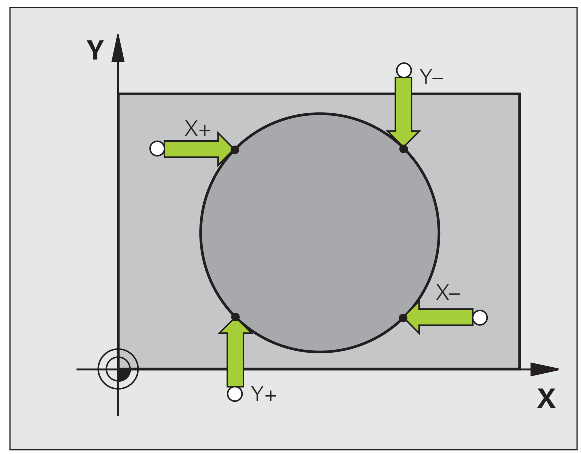
For incomplete circles (circular arcs) you can choose the appropriate probing direction.

- Position the touch probe approximately in the center of the circle.



- Select the probe function by pressing the PROBING CC soft key
- To probe the workpiece, press the machine START button four times. The touch probe touches four points on the inside of the circle.
- **Datum**: In the menu window, enter both coordinates of the circle center, confirm with the SET DATUM soft key, or write the values to a table (see "Writing the measured values from touch probe cycles in datum tables", page 343, or see "Writing the measured values from touch probe cycles in the preset table", page 343)
- To terminate the probe function, press the END soft key





Outside circle:

- Position the touch probe at a position near the first touch point outside of the circle.
- Select the probe direction by soft key.
- To probe the workpiece, press the machine START button.
- Repeat the probing process for the remaining three points. See figure at lower right.
- **Datum**: Enter the coordinates of the datum and confirm your entry with the SET DATUM soft key, or write the values to a table (see "Writing the measured values from touch probe cycles in datum tables", page 343, or see "Writing the measured values from touch probe cycles in the preset table", page 343)
- To terminate the probe function, press the END soft key

After the probing procedure is completed, the TNC displays the coordinates of the circle center and the circle radius PR

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

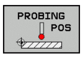
If you use probing functions in a tilted working plane, you must set 3-D ROT to **Active** for the Manual Operation and Automatic operating modes.

Measuring Workpieces with a 3-D Touch Probe

You can also use the touch probe in the Manual Operation and El. Handwheel operating modes to make simple measurements on the workpiece. Numerous programmable probe cycles are available for complex measuring tasks (see User's Manual, Cycles, Chapter 16, Checking workpieces automatically). With a 3-D touch probe you can determine:

- Position coordinates, and from them,
- Dimensions and angles on the workpiece.

To find the coordinate of a position on an aligned workpiece:



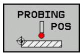
- ▶ Select the probe function by pressing the PROBING POS soft key
- ▶ Move the touch probe to a position near the touch point
- ▶ Select the probe direction and axis of the coordinate. Use the corresponding soft keys for selection
- ▶ To probe the workpiece, press the machine START button

The TNC shows the coordinates of the touch point as reference point

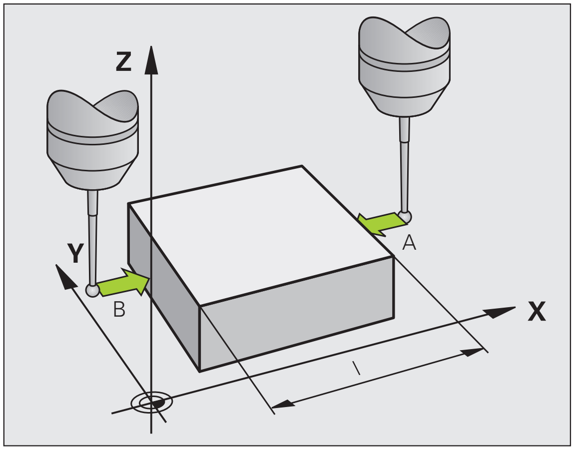
Finding the coordinates of a corner in the working plane

Find the coordinates of the corner point: See "Corner as datum" on page 351. The TNC displays the coordinates of the probed corner as reference point.

Measuring workpiece dimensions



- Select the probe function by pressing the PROBING POS soft key
- Position the touch probe at a position near the first touch point A
- Select the probing direction by soft key
- To probe the workpiece, press the machine START button
- If you will need the current datum later, write down the value that appears in the Datum display
- Datum: Enter "0"
- To terminate the dialog, press the END key
- Select the probe function by pressing the PROBING POS soft key
- Position the touch probe at a position near the second touch point B
- Select the probe direction with the soft keys: Same axis but from the opposite direction
- To probe the workpiece, press the machine START button



The value displayed as datum is the distance between the two points on the coordinate axis.

To return to the datum that was active before the length measurement:

- Select the probe function by pressing the PROBING POS soft key
- Probe the first touch point again.
- Set the datum to the value that you wrote down previously.
- To terminate the dialog, press the END key

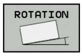
Measuring angles

You can use the 3-D touch probe to measure angles in the working plane. You can measure

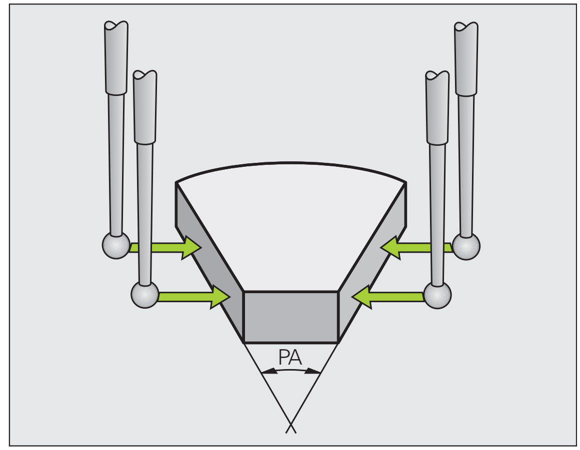
- the angle between the angle reference axis and a workpiece edge, or
- the angle between two sides

The measured angle is displayed as a value of maximum 90°.

Finding the angle between the angle reference axis and a workpiece edge

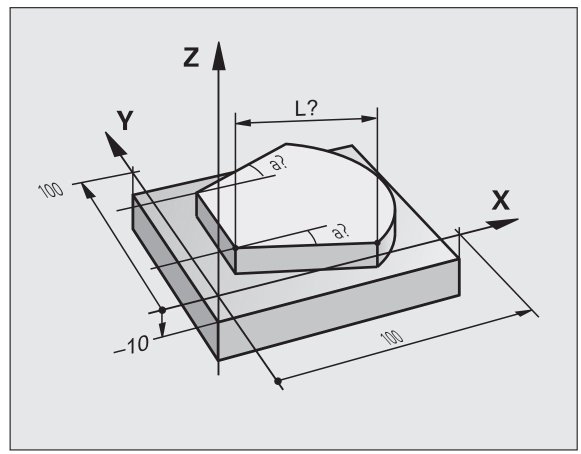


- Select the probe function by pressing the PROBING ROT soft key
- Rotation angle: If you need the current basic rotation later, write down the value that appears under Rotation angle
- Make a basic rotation with workpiece edge to be compared (see "Compensating Workpiece Misalignment with a 3-D Touch Probe" on page 348)
- Press the PROBING ROT soft key to display the angle between the angle reference axis and the workpiece edge as the rotation angle
- Cancel the basic rotation, or restore the previous basic rotation
- This is done by setting the rotation angle to the value that you previously wrote down.



To measure the angle between two workpiece sides:

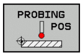
- Select the probe function by pressing the PROBING ROT soft key
- Rotation angle: If you need the current basic rotation later, write down the displayed rotation angle.
- Make a basic rotation with first workpiece edge (see "Compensating Workpiece Misalignment with a 3-D Touch Probe" on page 348)
- Probe the second edge as for a basic rotation, but do not set the rotation angle to zero!
- Press the PROBING ROT soft key to display the angle PA between the sides as the rotation angle.
- Cancel the basic rotation, or restore the previous basic rotation by setting the rotation angle to the value that you wrote down previously.



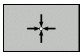
Using the touch probe functions with mechanical probes or dial gauges

If you do not have an electronic 3-D touch probe on your machine, you can also use all the previously described manual touch probe functions (exception: calibration function) with mechanical probes or by simply touching the workpiece with the tool.

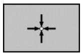
In place of the electronic signal generated automatically by a 3-D touch probe during probing, you can manually initiate the trigger signal for capturing the **probing position** by pressing a key. Proceed as follows:



- Select any touch probe function by soft key
- Move the mechanical probe to the first position to be captured by the TNC



- Confirm the position: Press the actual-position-capture soft key for the TNC to save the current position
- Move the mechanical probe to the next position to be captured by the TNC

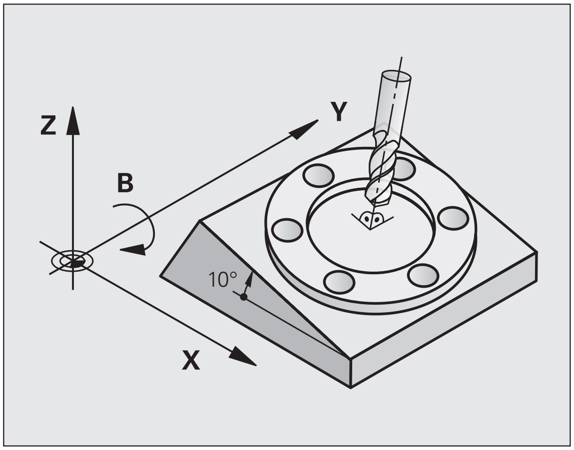


- Confirm the position: Press the actual-position-capture soft key for the TNC to save the current position
- If required, move to additional positions and capture as described previously
- **Datum:** In the menu window, enter the coordinates of the new datum, confirm with the SET DATUM soft key, or write the values to a table (see "Writing the measured values from touch probe cycles in datum tables", page 343, or see "Writing the measured values from touch probe cycles in the preset table", page 343)
- To terminate the probe function, press the END key

12.9 Tilting the Working Plane (Software Option 1)

Application, function

The functions for tilting the working plane are interfaced to the TNC and the machine tool by the machine tool builder. With some swivel heads and tilting tables, the machine tool builder determines whether the entered angles are interpreted as coordinates of the rotary axes or as angular components of a tilted plane. Refer to your machine manual.



The TNC supports the tilting functions on machine tools with swivel heads and/or tilting tables. Typical applications are, for example, oblique holes or contours in an oblique plane. The working plane is always tilted around the active datum. The program is written as usual in a main plane, such as the X/Y plane, but is executed in a plane that is tilted relative to the main plane.

There are three functions available for tilting the working plane:

- Manual tilting with the3-D ROT soft key in the Manual Operation mode and Electronic Handwheel mode, see "Activating manual tilting", page 360
- Tilting under program control, Cycle **G80** in the part program (see User's Manual, Cycles, Cycle 19 WORKING PLANE)
- Tilting under program control, **PLANE** function in the part program (see "The PLANE Function: Tilting the Working Plane (Software Option 1)" on page 299)

The TNC functions for "tilting the working plane" are coordinate transformations. The working plane is always perpendicular to the direction of the tool axis.

When tilting the working plane, the TNC differentiates between two machine types:

- **Machine with tilting table**
  - You must tilt the workpiece into the desired position for machining by positioning the tilting table, for example with an L block.
  - The position of the transformed tool axis **does not change** in relation to the machine-based coordinate system. Thus if you rotate the table—and therefore the workpiece—by 90° for example, the coordinate system **does not rotate.** If you press the Z+ axis direction button in the Manual Operation mode, the tool moves in Z+ direction.
  - In calculating the transformed coordinate system, the TNC considers only the mechanically influenced offsets of the particular tilting table (the so-called "translational" components).
- **Machine with swivel head**
  - You must bring the tool into the desired position for machining by positioning the swivel head, for example with an L block.
  - The position of the transformed tool axis changes in relation to the machine-based coordinate system. Thus if you rotate the swivel head of your machine—and therefore the tool—in the B axis by 90° for example, the coordinate system rotates also. If you press the Z+ axis direction button in the Manual Operation mode, the tool moves in X+ direction of the machine-based coordinate system.
  - In calculating the transformed coordinate system, the TNC considers both the mechanically influenced offsets of the particular swivel head (the so-called “translational” components) and offsets caused by tilting of the tool (3-D tool length compensation).

Traversing the reference points in tilted axes

The TNC automatically activates the tilted working plane if this function was enabled when the control was switched off. Then the TNC moves the axes in the tilted coordinate system when an axis-direction key is pressed. Position the tool in such a way that a collision is excluded during the subsequent crossing of the reference points. To cross the reference points you have to deactivate the "Tilt Working Plane" function, see "Activating manual tilting", page 360.

Danger of collision!

Be sure that the function for tilting the working plane is active in the Manual Operation mode and that the angle values entered in the menu match the actual angles of the tilted axis.

Deactivate the "Tilt Working Plane" function before you cross the reference points. Take care that there is no collision. Retract the tool from the current position first, if necessary.

Position display in a tilted system

The positions displayed in the status window **(ACTL.** and **NOML.)** are referenced to the tilted coordinate system.

Limitations on working with the tilting function

- The probing function for basic rotation is not available if you have activated the working plane function in the Manual Operation mode.
- The actual-position-capture function is not allowed if the tilted working plane function is active.
- PLC positioning (determined by the machine tool builder) is not possible.

Activating manual tilting

To select manual tilting, press the 3-D ROT soft key

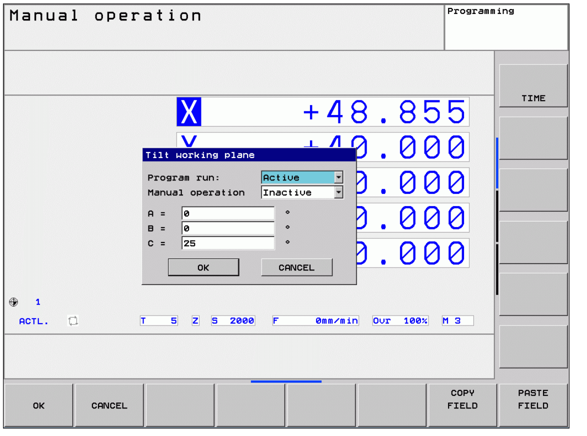
Use the arrow keys to move the highlight to the **Manual Operation** menu item

To activate manual tilting, press the ACTIVE soft key

Use the arrow keys to position the highlight on the desired rotary axis

Enter the tilt angle

To conclude entry, press the END key



To reset the tilting function, set the desired operating modes in the menu "Tilt working plane" to inactive.

If the tilted working plane function is active and the TNC moves the machine axes in accordance with the tilted axes, the status display shows the symbol.

If you activate the "Tilt working plane" function for the Program Run operating mode, the tilt angle entered in the menu becomes active in the first block of the part program. If you use Cycle **G80** or the **PLANE** function in the part program, the angle values defined there are in effect. Angle values entered in the menu will be overwritten.

13

Positioning with Manual Data Input

13.1 Programming and Executing Simple Machining Operations

The Positioning with Manual Data Input mode of operation is particularly convenient for simple machining operations or to pre-position the tool. It enables you to write a short program in HEIDENHAIN conversational programming or in ISO format, and execute it immediately. You can also call TNC cycles. The program is stored in the file $MDI. In the Positioning with MDI mode of operation, the additional status displays can also be activated.

Positioning with Manual Data Input (MDI)

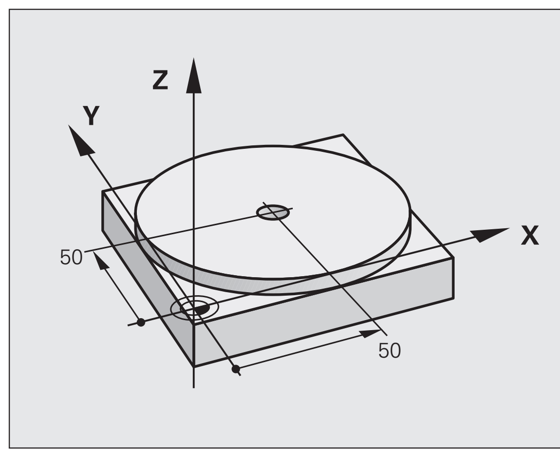
Limitation

The following functions are not available in the MDI mode:

- FK free contour programming
- Program section repeats
- Subprogramming
- Path compensation
- The programming graphics
- Program call %
- The program-run graphics

Select the Positioning with MDI mode of operation. Program the file $MDI as you wish

To start program run, press the machine START key



Example 1

A hole with a depth of 20 mm is to be drilled into a single workpiece. After clamping and aligning the workpiece and setting the datum, you can program and execute the drilling operation in a few lines.

First you pre-position the tool with straight-line blocks to the hole center coordinates at a setup clearance of 5 mm above the workpiece surface. Then drill the hole with Cycle **G200**.

%$MDI G71 *	
N10 T1 G17 S2000 *	Call tool: tool axis Z
	Spindle speed 2000 rpm
N20 G00 G40 G90 Z+200 *	Retract tool (rapid traverse)
N30 X+50 Y+50 M3 *	Move the tool at rapid traverse to a position above the hole
	Spindle on
N40 G01 Z+2 F2000 *	Position tool to 2 mm above the hole
N50 G200 DRILLING *	Define Cycle G200 Drilling
Q200=2 ;SETUP CLEARANCE	Setup clearance of the tool above the hole
Q201=-20 ;DEPTH	Hole depth (algebraic sign=working direction)
Q206=250 ;FEED RATE FOR PLNGN	Feed rate for drilling
Q202=10 ;PLUNGING DEPTH	Depth of each infeed before retraction
Q210=0 ;DWELL TIME AT TOP	Dwell time at top for chip release (in seconds)
Q203=+0 ;SURFACE COORDINATE	Workpiece surface coordinate
Q204=50 ;2ND SET-UP CLEARANCE	Position after the cycle, with respect to Q203
Q211=0.5 ;DWELL TIME AT DEPTH	Dwell time in seconds at the hole bottom
N60 G79 *	Call Cycle G200 PECKING
N70 G00 G40 Z+200 M2 *	Retract the tool
N9999999 %$MDI G71 *	End of program

Straight-line function: See "Straight line at rapid traverse G00 Straight line with feed rate G01 F" on page 164, DRILLING cycle: See User’s Manual, Cycles, Cycle 200 DRILLING.

Example 2: Correcting workpiece misalignment on machines with rotary tables

Use the 3-D touch probe to rotate the coordinate system. See "Touch Probe Cycles in the Manual Operation and El. Handwheel modes of operation," section "Compensating workpiece misalignment," in the Touch Probe Cycles User's Manual.

Write down the rotation angle and cancel the basic rotation

Select operating mode: Positioning with MDI

Select the rotary table axis, enter the rotation angle and feed rate you wrote down, for example: **G01 G40 G90 C+2.561 F50**

Conclude entry

Press the machine START button: The rotation of the table corrects the misalignment

Protecting and erasing programs in $MDI

The $MDI file is generally intended for short programs that are only needed temporarily. Nevertheless, you can store a program, if necessary, by proceeding as described below:

(Programming and Editing key)	Select the Programming and Editing mode of operation
PGM MGT	Press the PGM MGT key (program management) to call the file manager
↑	Mark the $MDI file
COPY ABC → XYZ	To select the file copying function, press the COPY soft key

DESTINATION FILE =

HOLE	Enter the name under which you want to save the current contents of the $MDI file
EXECUTE	Copy the file
END	To close the file manager, press the END soft key

For more information: see "Copying a single file", page 98.

14

Test Run and Program Run

14.1 Graphics

Application

In the program run modes of operation as well as in the Test Run mode, the TNC graphically simulates the machining of the workpiece. Using soft keys, select whether you desire:

- Plan view
- Projection in three planes
- 3-D view

The TNC graphic depicts the workpiece as if it were being machined with a cylindrical end mill. If a tool table is active, you can also simulate the machining operation with a spherical cutter. For this purpose, enter R2 = R in the tool table.

The TNC will not show a graphic if

- the current program has no valid blank form definition
- no program is selected

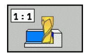
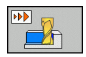
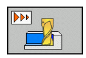
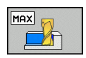
The TNC graphic does not show a radius oversize **DR** that has been programmed in the **T** block.

A graphic simulation is only possible under certain conditions for program sections or programs in which rotary axis movements are defined. The graphic may not be displayed correctly by the TNC.

Setting the speed of the test run

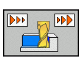
The most recently set speed remains active, even if the power is interrupted, until you change it.

After you have started a program, the TNC displays the following soft keys with which you can set the simulation speed.

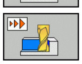
Functions	Soft key
Execute test run at the same speed at which the program will be run (programmed feed rates are taken into account).	
Increase the test speed incrementally.	
Decrease the test speed incrementally.	
Test run at the maximum possible speed (default setting).	

You can also set the simulation speed before you start a program:

- Switch to the next soft-key row



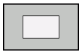
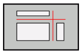
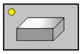
- Select the function for setting the simulation speed



- Select the desired function by soft key, e.g. incrementally increasing the test speed

Overview of display modes

The TNC displays the following soft keys in the Program Run and Test Run modes of operation:

View	Soft key
Plan view	
Projection in three planes	
3-D view	

Limitations during program run

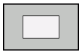
A graphical representation of a running program is not possible if the microprocessor of the TNC is already occupied with complicated machining tasks or if large areas are being machined. Example: Multipass milling over the entire blank form with a large tool. The TNC interrupts the graphics and displays the text **ERROR** in the graphics window. The machining process is continued, however.

In the test run graphics, the TNC does not depict multi-axis operations during machining. The error message **Axis cannot be shown** appears in the graphics window in such cases.

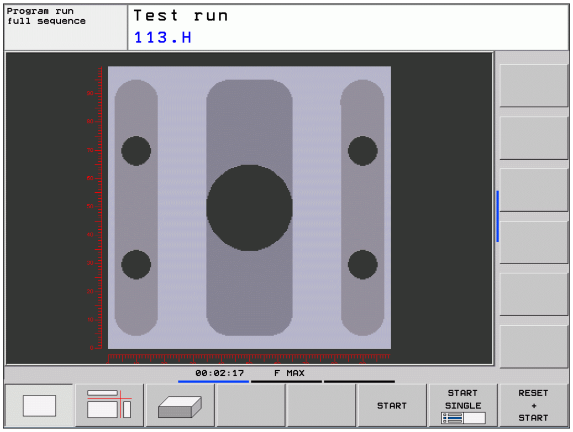
Plan view

This is the fastest of the graphic display modes.

If your machine has a mouse, the status bar shows the depth of any location on the workpiece when you move the mouse pointer over it.



- Press the soft key for plan view
- Regarding depth display, remember: The deeper the surface, the darker the shade

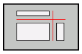


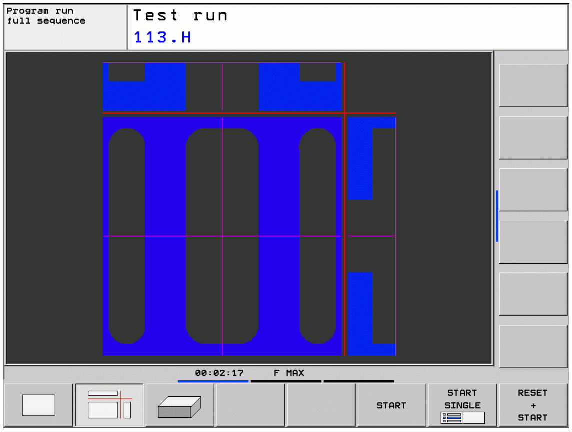
Projection in 3 planes

Similar to a workpiece drawing, the part is displayed with a plan view and two sectional planes. A symbol to the lower left indicates whether the display is in first angle or third angle projection according to ISO 5456-2 (selected with MP7310).

Details can be isolated in this display mode for magnification (see "Magnifying details", page 374).

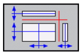
In addition, you can shift the sectional planes with the corresponding soft keys:



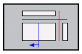
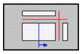
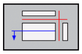
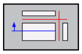
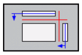
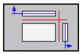


- Select the soft key for projection in three planes

- Shift the soft-key row until the soft key for the functions for shifting the sectional plane appears



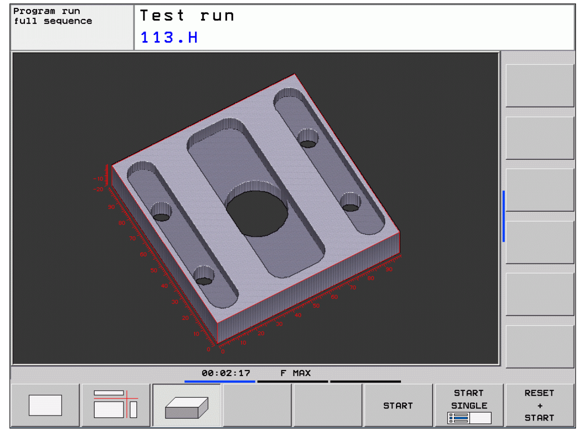
- Select the functions for shifting the sectional plane. The TNC offers the following soft keys:

Function	Soft keys	
Shift the vertical sectional plane to the right or left		
Shift the vertical sectional plane forward or backward		
Shift the horizontal sectional plane upwards or downwards		

The positions of the sectional planes are visible during shifting.

The default setting of the sectional plane is selected such that it lies in the working plane in the workpiece center and in the tool axis on the top surface.

3-D view

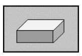


The workpiece is displayed in three dimensions.

You can rotate the 3-D display about the vertical and horizontal axes via soft keys. If there is a mouse attached to your TNC, you can also perform this function by holding down the right mouse button and dragging the mouse.

The shape of the workpiece can be depicted by a frame overlay at the beginning of the graphic simulation.

In the Test Run mode of operation you can isolate details for magnification, see "Magnifying details", page 374.

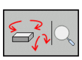


▶ Press the soft key for 3-D view

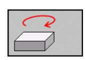
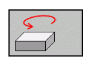
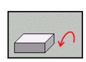
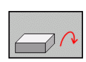
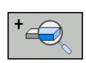
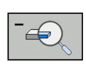
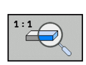
The speed of the 3-D graphics depends on the tooth length (**LCUTS** column in the tool table). If **LCUTS** is defined as 0 (basic setting), the simulation calculates an infinitely long tooth length, which leads to a long processing time.

Rotating and magnifying/reducing the 3-D view

▶ Shift the soft-key row until the soft key for the rotating and magnification/reduction appears



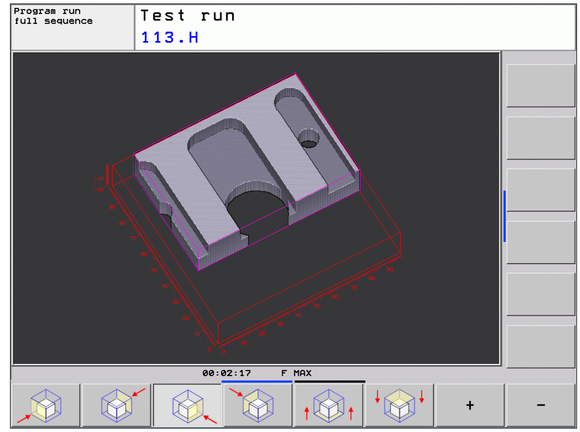
▶ Select functions for rotating and magnifying/reducing:

Function	Soft keys	
Rotate in 5° steps about the vertical axis		
Tilt in 5° steps about the horizontal axis		
Magnify the graphic stepwise. If the view is magnified, the TNC shows the letter **Z** in the footer of the graphic window.		
Reduce the graphic stepwise. If the view is reduced, the TNC shows the letter **Z** in the footer of the graphic window.		
Reset image to programmed size		

If there is a mouse attached to your TNC, you can also perform the functions described above with the mouse.

- In order to rotate the graphic shown in three dimensions: Hold the right mouse button down and move the mouse. After you release the right mouse button, the TNC orients the workpiece to the defined orientation
- In order to shift the graphic shown: Hold the center mouse button or the wheel button down and move the mouse. The TNC shifts the workpiece in the corresponding direction. After you release the center mouse button, the TNC shifts the workpiece to the defined position
- In order to zoom in on a certain area with the mouse: Draw a rectangular zoom area while holding the left mouse button down. After you release the left mouse button, the TNC zooms in on the defined area of the workpiece
- In order to quickly zoom in and out with the mouse: Rotate the wheel button forward or backward

Magnifying details



You can magnify details in all display modes in the Test Run mode and a Program Run mode.

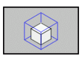
The graphic simulation or the program run, respectively, must first have been stopped. A detail magnification is always effective in all display modes.

Changing the detail magnification

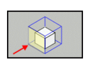
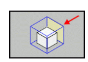
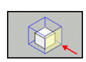
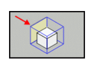
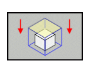
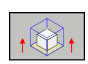
The soft keys are listed in the table

- Interrupt the graphic simulation, if necessary
- Shift the soft-key row in the Test Run mode, or in a Program Run mode, respectively, until the soft key for detail enlargement appears

- Shift the soft-key row until the soft-key for the detail magnification functions appears



- Select the functions for detail magnification
- Press the corresponding soft key to select the workpiece surface (see table below)
- To reduce or magnify the blank form, press and hold the MINUS or PLUS soft key, respectively
- Restart the test run or program run by pressing the START soft key (RESET + START returns the workpiece to its original state)

Function	Soft keys	
Select the left/right workpiece surface		
Select the front/back workpiece surface		
Select the top/bottom workpiece surface		
Shift the sectional plane to reduce or magnify the blank form	–	+
Select the isolated detail	TRANSFER DETAIL	

After a new workpiece detail magnification is selected, the control "forgets" previously simulated machining operations. The TNC then displays machined areas as unmachined areas.

If the workpiece blank cannot be further enlarged or reduced, the TNC displays an error message in the graphics window. To clear the error message, reduce or enlarge the workpiece blank.

Repeating graphic simulation

A part program can be graphically simulated as often as desired, either with the complete workpiece or with a detail of it.

Function	Soft key
Restore workpiece to the detail magnification in which it was last shown	RESET BLK FORM
Reset detail magnification so that the machined workpiece or workpiece blank is displayed as it was programmed with BLK FORM.	WINDOW BLK FORM

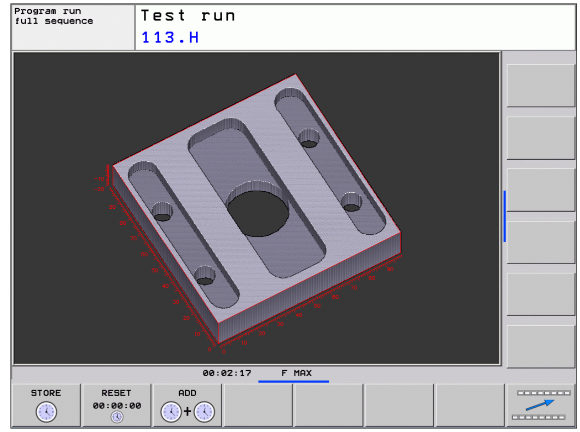
With the WINDOW BLK FORM soft key, you return the displayed workpiece blank to its originally programmed dimensions, even after isolating a detail without TRANSFER DETAIL.

Displaying the tool

You can display the tool during simulation in the plan view and in the projection in 3 planes. The TNC depicts the tool in the diameter defined in the tool table.

Function	Soft key
Do not display the tool during simulation	TOOLS DISPLAY HIDE
Display the tool during simulation	TOOLS DISPLAY HIDE

Measuring the machining time



Program Run modes of operation

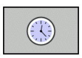
The timer counts and displays the time from program start to program end. The timer stops whenever machining is interrupted.

Test Run

The timer displays the time that the TNC calculates for the duration of tool movements that are executed at feed rate. Dwell times are included in the calculation by the TNC. The time calculated by the TNC can only conditionally be used for calculating the production time because the TNC does not account for the duration of machine-dependent interruptions, such as tool change.

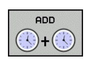
Activating the stopwatch function

- Shift the soft-key row until the soft-key for the stopwatch functions appears



- Select the stopwatch functions

- Select the desired function by soft key, e.g. saving the displayed time

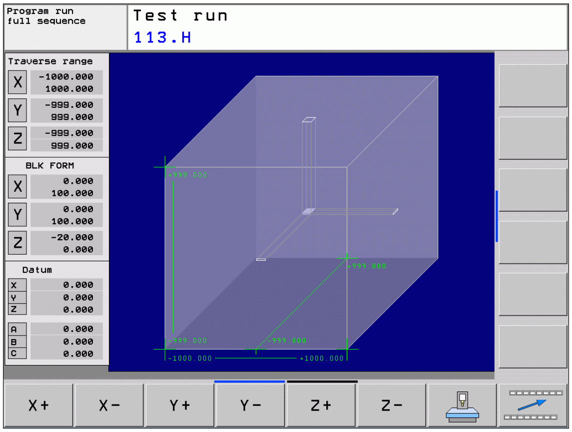
Stopwatch functions	Soft key
Store displayed time	
Display the sum of stored time and displayed time	
Clear displayed time	

During the Test Run, the TNC resets the machining time as soon as a new BLK form **G30/G3** is evaluated.

14.2 Showing the Blank in the Working Space

Application

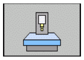
This MOD function enables you to graphically check the position of the workpiece blank or reference point in the machine's working space and to activate work space monitoring in the Test Run mode of operation. This function is activated with the **BLANK IN WORKSPACE** soft key. You can activate or deactivate the function with the **SW limit monitoring** soft key (2nd soft-key row).



Another transparent cuboid represents the workpiece blank. Its dimensions are shown in the **BLK FORM** table. The TNC takes the dimensions from the workpiece blank definition of the selected program. The workpiece cuboid defines the coordinate system for input. Its datum lies within the traverse-range cuboid.

For a test run it normally does not matter where the workpiece blank is located within the working space. However, if you activate working-space monitoring, you must graphically shift the workpiece blank so that it lies within the working space. Use the soft keys shown in the table.

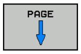
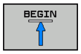
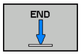
You can also activate the current datum for the Test Run operating mode (see the last line of the following table).

Function	Soft keys	
Shift workpiece blank in positive/negative X direction	X+	X-
Shift workpiece blank in positive/negative Y direction	Y+	Y-
Shift workpiece blank in positive/negative Z direction	Z+	Z-
Show workpiece blank referenced to the set datum		
Switch monitoring function on or off	SW limit monitoring	

14.3 Functions for Program Display

Overview

In the program run modes of operation as well as in the Test Run mode, the TNC provides the following soft keys for displaying a part program in pages:

Functions	Soft key
Go back in the program by one screen	
Go forward in the program by one screen	
Go to the beginning of the program	
Go to the end of the program	

14.4 Test Run

Application

In the Test Run mode of operation you can simulate programs and program sections to reduce programming errors during program run. The TNC checks the programs for the following:

- Geometrical incompatibilities
- Missing data
- Impossible jumps
- Violation of the machine's working space

The following functions are also available:

- Blockwise test run
- Interrupt test at any block
- Optional block skip
- Functions for graphic simulation
- Measuring the machining time
- Additional status display

Caution: Danger of collision!

The TNC cannot graphically simulate all traverse motions actually performed by the machine. These include

- Traverse motions during tool change, if the machine manufacturer defined them in a tool-change macro or via the PLC,
- Positioning movements that the machine manufacturer defined in an M-function macro,
- Positioning movements that the machine manufacturer performs via the PLC

HEIDENHAIN therefore recommends proceeding with caution for every new program, even when the program test did not output any error message, and no visible damage to the workpiece occurred.

After a tool call, the TNC always starts a program test at the following position:

- In the machining plane at the position X=0, Y=0
- In the tool axis, 1 mm above the **MAX** point defined in the **BLK FORM**

If you call the same tool, the TNC resumes program simulation from the position last programmed before the tool call.

In order to ensure unambiguous behavior during program run, after a tool change you should always move to a position from which the TNC can position the tool for machining without causing a collision.

Your machine tool builder can also define a tool-change macro for the Test Run operating mode. This macro will simulate the exact behavior of the machine. Please refer to your machine tool manual.

Executing a test run

If the central tool file is active, a tool table must be active (status S) to conduct a test run. Select a tool table via the file manager (PGM MGT) in the Test Run mode of operation.

With the BLANK IN WORK SPACE function, you activate work space monitoring for the test run, see "Showing the Blank in the Working Space", page 377.

- Select the Test Run operating mode
- Call the file manager with the PGM MGT key and select the file you wish to test, or
- Go to the program beginning: Select line 0 with the GOTO key and confirm your entry with the ENT key.

The TNC then displays the following soft keys:

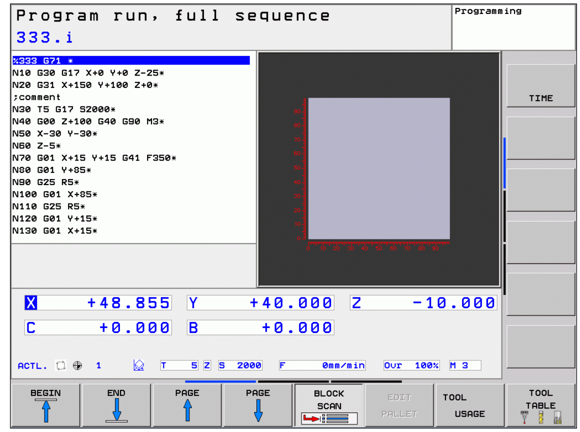
Functions	Soft key
Reset the blank form and test the entire program	RESET + START
Test the entire program	START
Test each program block individually	START SINGLE
Halt test run (soft key only appears once you have started the test run)	STOP

You can interrupt the test run and continue it again at any point—even within a fixed cycle. In order to continue the test, the following actions must not be performed:

- Selecting another block with the arrow keys or the GOTO key
- Making changes to the program
- Switching the operating mode
- Selecting a new program

14.5 Program Run

Application



In the Program Run, Full Sequence mode of operation the TNC executes a part program continuously to its end or up to a program stop.

In the Program Run, Single Block mode of operation you must start each block separately by pressing the machine START button.

The following TNC functions are available in the program run modes of operation:

- Interrupt program run
- Start program run from a certain block
- Optional block skip
- Editing the tool table TOOL.T
- Check and change Q parameters
- Superimpose handwheel positioning
- Functions for graphic simulation
- Additional status display

Running a part program

Preparation

1. Clamp the workpiece to the machine table
2. Set the datum
3. Select the necessary tables and pallet files (status M)
4. Select the part program (status M)

You can adjust the feed rate and spindle speed with the override knobs.

It is possible to reduce the feed rate when starting the NC program using the FMAX soft key. The reduction applies to all rapid traverse and feed rate movements. The value you enter is no longer in effect after the machine has been turned off and on again. In order to re-establish the respectively defined maximum feed rate after switch-on, you need to re-enter the corresponding value.

Program Run, Full Sequence

- Start the part program with the machine START button

Program Run, Single Block

- Start each block of the part program individually with the machine START button

Interrupting machining

There are several ways to interrupt a program run:

- Programmed interruptions
- Pressing the machine STOP button
- Switching to Program Run, Single Block

If the TNC registers an error during program run, it automatically interrupts the machining process.

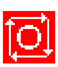
Programmed interruptions

You can program interruptions directly in the part program. The TNC interrupts the program run at a block containing one of the following entries:

- **G38** (with and without miscellaneous function)
- Miscellaneous functions **M0**, **M2** or **M30**
- Miscellaneous function **M6** (defined by the machine tool builder)

Interruption through the machine STOP button

- Press the machine STOP button: The block that the TNC is currently executing is not completed. The NC stop signal in the status display blinks (see table)
- If you do not wish to continue the machining process, you can reset the TNC with the INTERNAL STOP soft key. The NC stop signal in the status display goes out. In this case, the program must be restarted from the program beginning

Symbol	Meaning
	Program run is stopped

Interrupting the machining process by switching to the Program Run, Single Block mode of operation

You can interrupt a program that is being run in the Program Run, Full Sequence mode of operation by switching to the Program Run, Single Block mode. The TNC interrupts the machining process at the end of the current block.

Moving the machine axes during an interruption

You can move the machine axes during an interruption in the same way as in the Manual Operation mode.

Application example:
Retracting the spindle after tool breakage

- Interrupt machining
- Enable the external direction keys: Press the MANUAL TRAVERSE soft key
- Move the axes with the machine axis direction buttons

On some machines you may have to press the machine START button after the MANUAL OPERATION soft key to enable the axis direction buttons. Refer to your machine manual.

Resuming program run after an interruption

If you cancel a program with INTERNAL STOP, you have to start the program with the RESTORE POS. AT N function or with GOTO "0".

If a program run is interrupted during a fixed cycle, the program must be resumed from the beginning of the cycle. This means that some machining operations will be repeated.

If you interrupt a program run during execution of a subprogram or program section repeat, use the RESTORE POS AT N function to return to the position at which the program run was interrupted.

When a program run is interrupted, the TNC stores:

- The data of the last defined tool
- Active coordinate transformations (e.g. datum shift, rotation, mirroring)
- The coordinates of the circle center that was last defined

Note that the stored data remain active until they are reset (e.g. if you select a new program).

The stored data are used for returning the tool to the contour after manual machine axis positioning during an interruption (RESTORE POSITION soft key).

Resuming program run with the START button

You can resume program run by pressing the machine START button if the program was interrupted in one of the following ways:

- The machine STOP button was pressed
- Programmed interruption

Resuming program run after an error

If the error message is not blinking:

- Remove the cause of the error
- To clear the error message from the screen, press the CE key
- Restart the program, or resume program run where it was interrupted

If the error message is blinking:

- Press and hold the END key for two seconds. This induces a TNC system restart
- Remove the cause of the error
- Start again

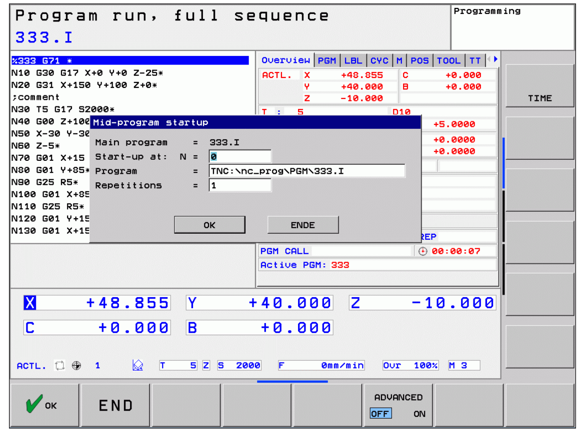
If you cannot correct the error, write down the error message and contact your repair service agency.

Mid-program startup (block scan)

The RESTORE POS AT N feature must be enabled and adapted by the machine tool builder. Refer to your machine manual.

With the RESTORE POS AT N feature (block scan) you can start a part program at any block you desire. The TNC scans the program blocks up to that point. Machining can be graphically simulated.

If you have interrupted a part program with an INTERNAL STOP, the TNC automatically offers the interrupted block N for mid-program startup.



Mid-program startup must not begin in a subprogram.

All necessary programs, tables and pallet files must be selected in a program run mode of operation (status M).

If the program contains a programmed interruption before the startup block, the block scan is interrupted. Press the machine START button to continue the block scan.

After a block scan, return the tool to the calculated position with RESTORE POSITION.

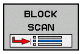
Tool length compensation does not take effect until after the tool call and a following positioning block. This also applies if you have only changed the tool length.

The TNC skips all touch probe cycles in a mid-program startup. Result parameters that are written to from these cycles might therefore remain empty.

You may not use mid-program startup if the following occurs after a tool change in the machining program:

- The program is started in an FK sequence
- The stretch filter is active
- Pallet management is used
- The program is started in a threading cycle (Cycles 17, 18, 19, 206, 207 and 209) or the subsequent program block
- Touch-probe cycles 0, 1 and 3 are used before program start

- Go to the first block of the current program to start a block scan: Enter GOTO "0"



- To select block scan, press the BLOCK SCAN soft key, or
- **`Start-up at N:`** Enter the block number N at which the block scan should end
- **`Program:`** Enter the name of the program containing block N
- **`Repetitions:`** If block N is located in a program section repeat or in a subprogram that is to be run repeatedly, enter the number of repetitions to be calculated in the block scan
- To start the block scan, press the machine START button
- Contour approach (see following section)

Entering a program with the GOTO key

If you use the GOTO block number key for going into a program, neither the TNC nor the PLC will execute any functions that ensure a safe start.

If you use the GOTO block number key for going into a subprogram,

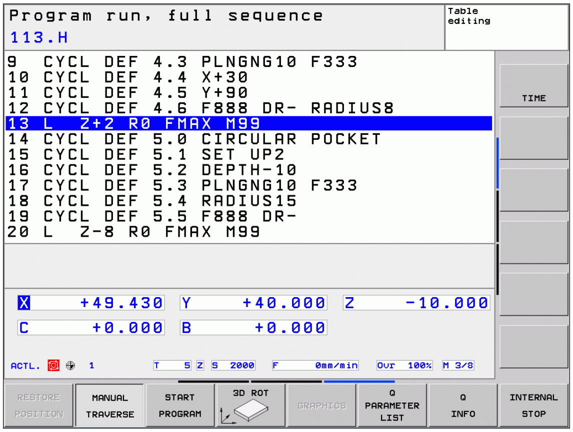
- the TNC will skip the end of the subprogram (**`G98 L0`**)
- the TNC will reset function M126 (Shorter-path traverse of rotary axes)

In such cases you must always use the mid-program startup function.

Returning to the contour

With the RESTORE POSITION function, the TNC returns to the workpiece contour in the following situations:

- Return to the contour after the machine axes were moved during a program interruption that was not performed with the INTERNAL STOP function.
- Return to the contour after a block scan with RESTORE POS AT N, for example after an interruption with INTERNAL STOP.
- Depending on the machine, if the position of an axis has changed after the control loop has been opened during a program interruption.



- ▶ To select a return to contour, press the RESTORE POSITION soft key
- ▶ Restore machine status, if required
- ▶ To move the axes in the sequence that the TNC suggests on the screen, press the machine START button
- ▶ To move the axes in any sequence, press the soft keys RESTORE X, RESTORE Z, etc., and activate each axis with the machine START button
- ▶ To resume machining, press the machine START button

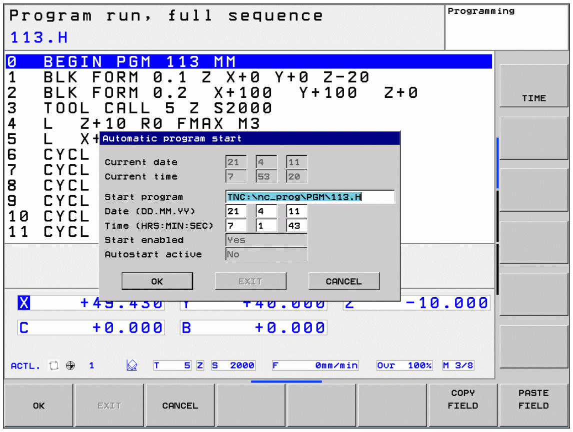
14.6 Automatic Program Start

Application

The TNC must be specially prepared by the machine tool builder for use of the automatic program start function. Refer to your machine manual.

Caution: Danger for the operator!

The autostart function must not be used on machines that do not have an enclosed working space.



In a Program Run operating mode, you can use the AUTOSTART soft key (see figure at upper right) to define a specific time at which the program that is currently active in this operating mode is to be started:

AUTOSTART

- Show the window for entering the starting time (see figure at center right)
- **Time (h:min:sec):** Time of day at which the program is to be started
- **Date (DD.MM.YYYY):** Date at which the program is to be started
- To activate the start, press the OK soft key

14.7 Optional Block Skip

Application

In a test run or program run, the control can skip over blocks that begin with a slash "/":

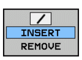
- To run or test the program without the blocks preceded by a slash, set the soft key to ON

- To run or test the program with the blocks preceded by a slash, set the soft key to OFF

> This function does not work for **`TOOL DEF`** blocks.
>
> After a power interruption the TNC returns to the most recently selected setting.

Insert the "/" character

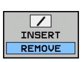
- In the **`Programming`** mode you select the block in which the character is to be inserted.



- Select the INSERT soft key

Erasing the "/" character

- In the **`Programming`** mode you select the block in which the character is to be deleted.



- Select the REMOVE soft key

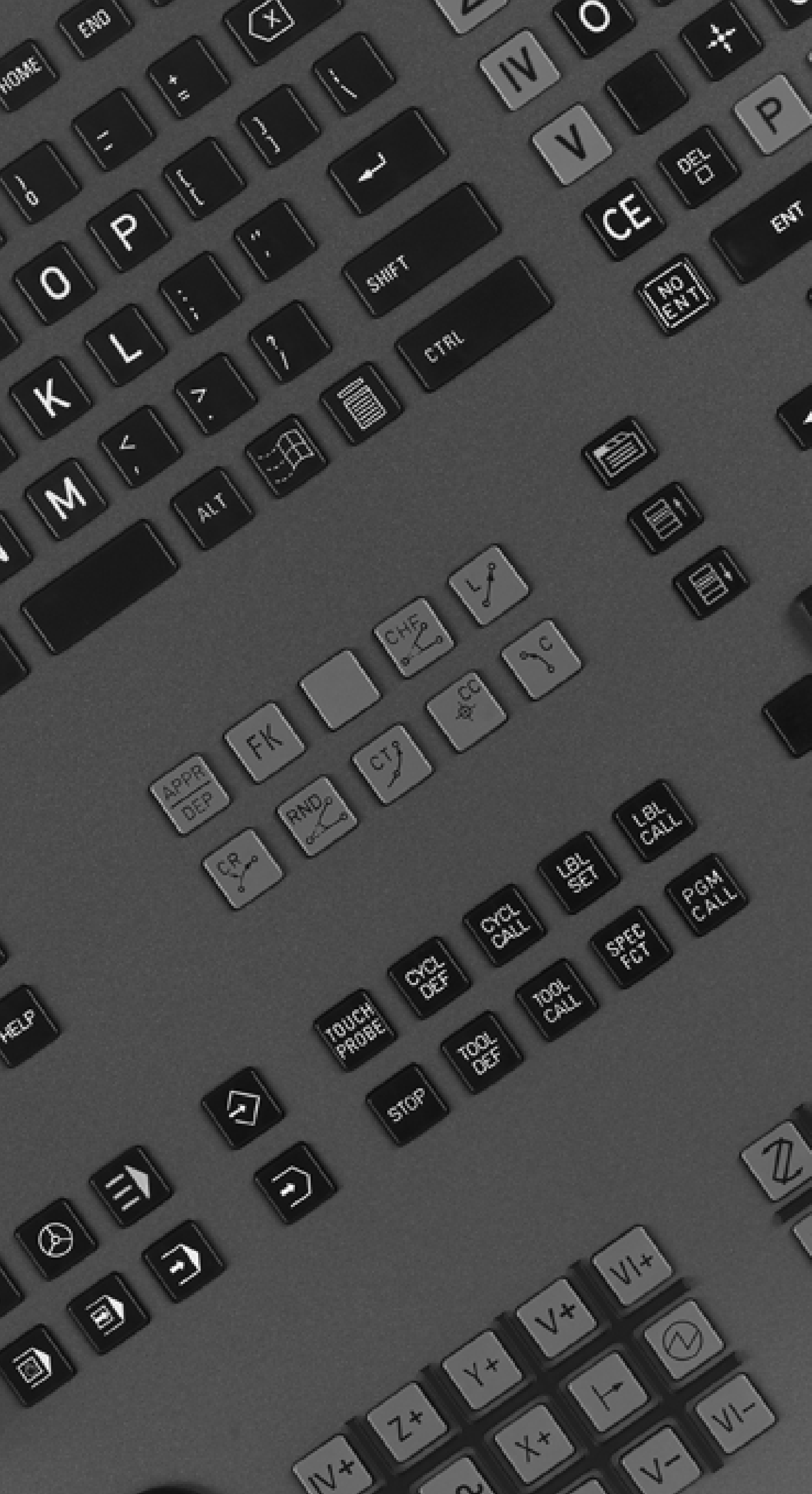
14.8 Optional Program-Run Interruption

Application

The TNC optionally interrupts program run at blocks containing M1. If you use M1 in the Program Run mode, the TNC does not switch off the spindle or coolant.

- Do not interrupt Program Run or Test Run at blocks containing M1: Set soft key to OFF

- Interrupt Program Run or Test Run at blocks containing M1: Set soft key to ON



15

MOD Functions

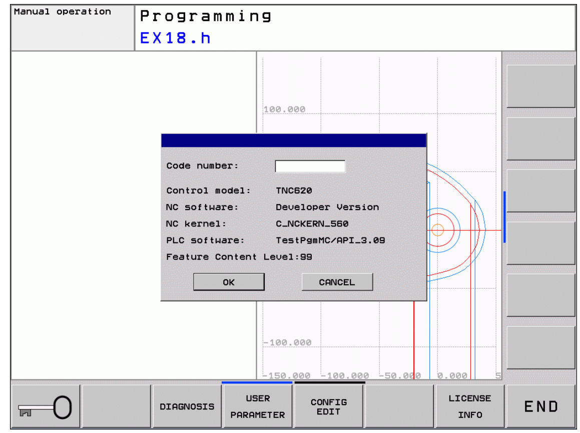
15.1 Selecting MOD Functions

The MOD functions provide additional input possibilities and displays. The available MOD functions depend on the selected operating mode.

Selecting the MOD functions

Call the operating mode in which you wish to change the MOD functions.

▶ To select the MOD functions, press the MOD key. The figures at right show typical screen menus in Programming and Editing mode (figure at upper right), Test Run mode (figure at lower right) and in a machine operating mode (see figure on next page)



Changing the settings

▶ Select the desired MOD function in the displayed menu with the arrow keys

There are three possibilities for changing a setting, depending on the function selected:

- Enter a numerical value directly, e.g. when determining the traverse range limit
- Change a setting by pressing the ENT key, e.g. when setting program input
- Change a setting via a selection window. If more than one possibility is available for a particular setting, you can superimpose a window listing all of the given possibilities by pressing the GOTO key. Select the desired setting directly by pressing the corresponding numerical key (to the left of the colon), or by using the arrow keys and then confirming with ENT. If you don't want to change the setting, close the window again with END

Exiting the MOD functions

▶ To exit the MOD functions, press the END key or END soft key

Overview of MOD functions

The functions available depend on the momentarily selected operating mode:

Programming:

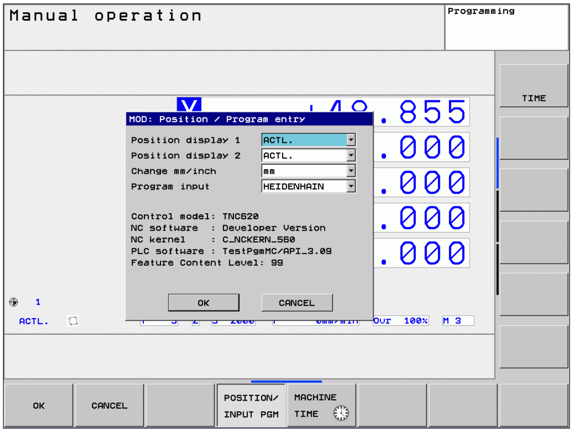
- Display software numbers
- Enter code number
- Machine-specific user parameters, if applicable
- Legal information

Test run:

- Display software numbers
- Show active tool table in the test run
- Show active datum table in the test run

In all other modes:

- Display software numbers
- Select position display
- Define unit of measurement (mm/inches)
- Set the programming language for MDI
- Selection of axes for actual position capture
- Display operating times



15.2 Software Numbers

Application

The following software numbers are displayed on the TNC screen after the MOD functions have been selected:

- `Control model`: Designation of the control (managed by HEIDENHAIN)
- **NC software**: Number of the NC software (managed by HEIDENHAIN)
- **NC software**: Number of the NC software (managed by HEIDENHAIN)
- **NC Kernel**: Number of the NC software (managed by HEIDENHAIN)
- **PLC software**: Number or name of the PLC software (managed by your machine tool builder)
- **Feature Content Level (FCL)**: Development level of the software installed on the control (see "Feature content level (upgrade functions)" on page 7)

15.3 Entering Code Numbers

Application

The TNC requires a code number for the following functions:

Function	Code number
Select user parameters	123
Configuring an Ethernet card	NET123
Enable special functions for Q parameter programming	555343

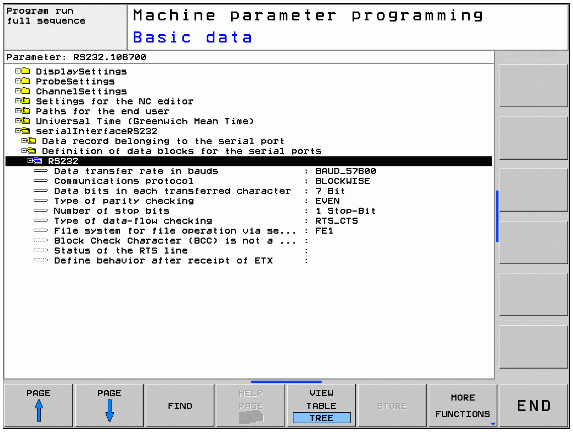
15.4 Setting the Data Interfaces

Serial interfaces on the TNC 320

The TNC 320 automatically uses the LSV2 transmission protocol for serial data transfer. The LSV2 protocol is permanent and cannot be changed except for setting the baud rate (machine parameter **baudRateLsv2**). You can also specify another type of transmission (interface). The settings described below are therefore effective only for the respective newly defined interface.

Application

To set up a data interface, select the file management (PGM MGT) and press the MOD key. Press the MOD key again and enter the code number 123. The TNC shows the user parameter **GfgSerialInterface**, in which you can enter the following settings:



Setting the RS-232 interface

Open the RS232 folder. The TNC then displays the following settings:

Setting the baud rate (baudRate)

You can set the BAUD RATE (data transfer speed) from 110 to 115 200 baud.

Set the protocol (protocol)

The data communication protocol controls the data flow of a serial transmission (comparable to MP5030 of the iTNC 530).

Here, the BLOCKWISE setting designates a form of data transfer where data is transmitted in blocks. This is not to be confused with the blockwise data reception and simultaneous blockwise processing by older TNC contouring controls. Blockwise reception of an NC program and simultaneous machining of the program is not possible!

Communications protocol	Selection
Standard data transfer	STANDARD
Packet-based data transfer	BLOCKWISE
Transmission without protocol	RAW_DATA

Set the data bits (dataBits)

By setting the data bits you define whether a character is transmitted with 7 or 8 data bits.

Parity check (parity)

The parity bit helps the receiver to detect transmission errors. The parity bit can be formed in three different ways:

- No parity (NONE): There is no error detection
- Even parity (EVEN): Here there is an error if the receiver finds that it has received an odd number of set bits
- Odd parity (ODD): Here there is an error if the receiver finds that it has received an even number of set bits

Setting the stop bits (stopBits)

The start bit and one or two stop bits enable the receiver to synchronize to every transmitted character during serial data transmission.

Setting the handshake (flowControl)

By handshaking, two devices control data transfer between them. A distinction is made between software handshaking and hardware handshaking.

- No data flow checking (NONE): Handshaking is not active
- Hardware handshaking (RTS_CTS): Transmission stop is active through RTS
- Software handshaking (XON_XOFF): Transmission stop is active through DC3 (XOFF)

Settings for data transfer with the TNCserver PC software

Enter the following settings in the user parameters (**serialInterfaceRS232 / definition of data blocks for the serial ports / RS232**):

Parameter	Selection
Data transfer rate in baud	Has to match the setting in TNCserver
Communications protocol	BLOCKWISE
Data bits in each transferred character	7 bits
Type of parity checking	EVEN
Number of stop bits	1 stop bit
Specify type of handshake:	RTS_CTS
File system for file operations	FE1

Setting the operating mode of the external device (fileSystem)

The functions "Transfer all files," "Transfer selected file," and "Transfer directory" are not available in the FE2 and FEX modes.

External device	Operating mode	Symbol
PC with HEIDENHAIN data transfer software TNCremoNT	LSV2	
HEIDENHAIN floppy disk units	FE1	
Non-HEIDENHAIN devices such as printers, scanners, punchers, PC without TNCremoNT	FEX	

Software for data transfer

For transfer of files to and from the TNC, we recommend using the HEIDENHAIN TNCremo data transfer software. With TNCremo, data transfer is possible with all HEIDENHAIN controls via the serial interface or the Ethernet interface.

> You can download the current version of TNCremo free of charge from the HEIDENHAIN Filebase (www.heidenhain.de, <Services and Documentation>, <Software>, <PC Software>, <TNCremoNT>).

System requirements for TNCremo:

- PC with 486 processor or higher
- Windows 95, Windows 98, Windows NT 4.0, Windows 2000, Windows XP or Windows Vista operating systems
- 16 MB RAM
- 5 MB free memory space on your hard disk
- An available serial interface or connection to the TCP/IP network

Installation under Windows

- Start the SETUP.EXE installation program with the File Manager (Explorer)
- Follow the setup program instructions

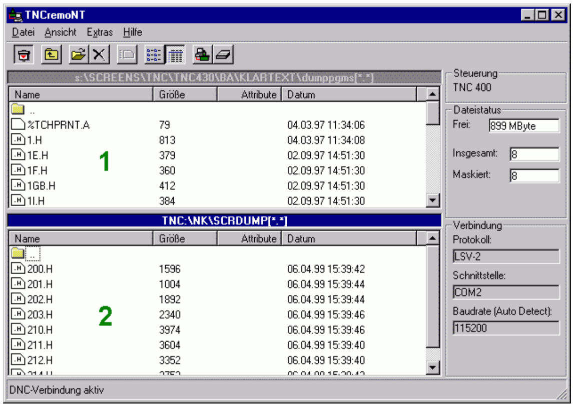
Starting TNCremo under Windows

- Click <Start>, <Programs>, <HEIDENHAIN Applications>, <TNCremo>

When you start TNCremo for the first time, TNCremo automatically tries to set up a connection with the TNC.

Data transfer between the TNC and TNCremoNT

Before you transfer a program from the TNC to the PC, you must make absolutely sure that you have already saved the program currently selected on the TNC. The TNC saves changes automatically when you switch the mode of operation on the TNC, or when you select the file manager via the PGM MGT key.



Check whether the TNC is connected to the correct serial port on your PC or to the network.

Once you have started TNCremoNT, you will see a list of all files that are stored in the active directory in the upper section of the main window **1**. Using the menu items <File> and <Change directory>, you can change the active directory or select another directory on your PC.

If you want to control data transfer from the PC, establish the connection with your PC in the following manner:

- Select <File>, <Setup connection>. TNCremoNT now receives the file and directory structure from the TNC and displays this at the bottom left of the main window **2**
- To transfer a file from the TNC to the PC, select the file in the TNC window with a mouse click and drag and drop the highlighted file into the PC window **1**
- To transfer a file from the PC to the TNC, select the file in the PC window with a mouse click and drag and drop the highlighted file into the TNC window **2**

If you want to control data transfer from the TNC, establish the connection with your PC in the following manner:

- Select <Extras>, <TNCserver>. TNCremoNT is now in server mode. It can receive data from the TNC and send data to the TNC
- You can now call the file management functions on the TNC by pressing the PGM MGT key (see "Data transfer to or from an external data medium" on page 104) and transfer the desired files

Exiting TNCremoNT

Select the menu items <File>, <Exit>

Refer also to the TNCremoNT context-sensitive help texts where all of the functions are explained in more detail. The help texts must be called with the F1 key.

15.5 Ethernet Interface

Introduction

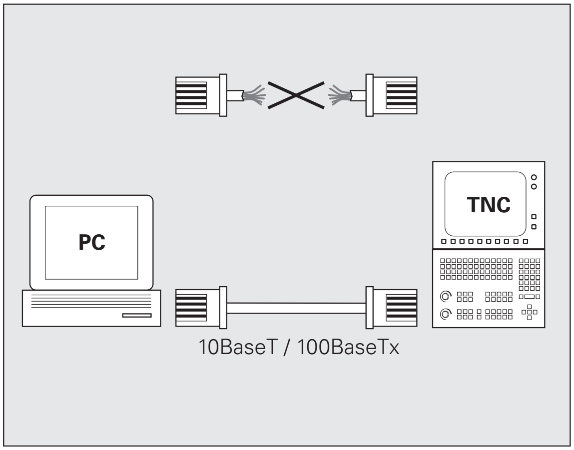
The TNC is shipped with a standard Ethernet card to connect the control as a client in your network. The TNC transmits data via the Ethernet card with

- the **smb** protocol (**s**erver **m**essage **b**lock) for Windows operating systems, or
- the **TCP/IP** protocol family (Transmission Control Protocol/Internet Protocol) and with support from the NFS (Network File System).

Connection possibilities

You can connect the Ethernet card in your TNC to your network through the RJ45 connection (X26, 100BaseTX or 10BaseT), or directly to a PC. The connection is metallically isolated from the control electronics.

For a 100BaseTX or 10BaseT connection you need a Twisted Pair cable to connect the TNC to your network.



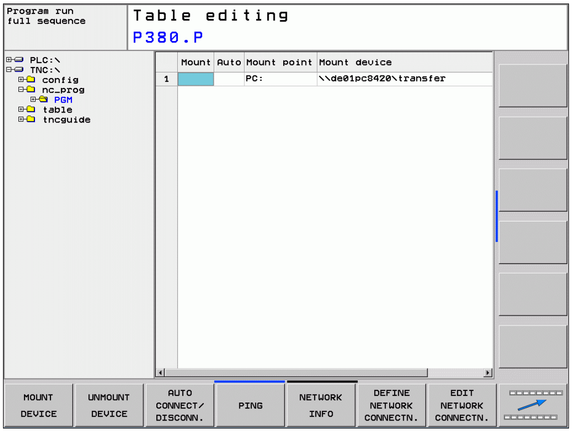
The maximum cable length between TNC and a node depends on the quality grade of the cable, the sheathing and the type of network (100BaseTX or 10BaseT).

No great effort is required to connect the TNC directly to a PC that has an Ethernet card. Simply connect the TNC (port X26) and the PC with an Ethernet crossover cable (trade names: crossed patch cable or STP cable).

Connecting the control to the network

Function overview of network configuration

- In the file manager (PGM MGT), press the **Network** soft key

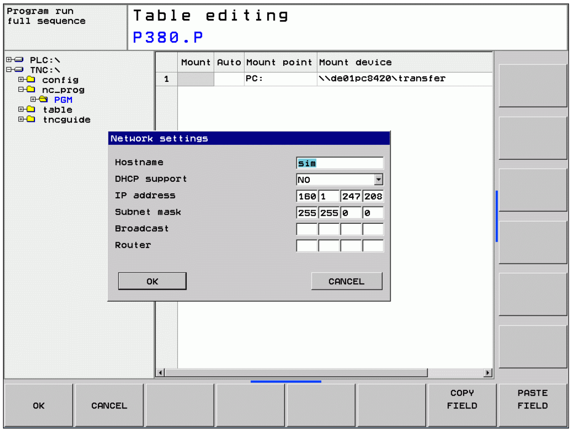


Function	Soft key
Establishes the connection to the selected network drive. Successful connection is indicated by a check mark under Mount.	MOUNT DEVICE
Separates the connection to a network drive.	UNMOUNT DEVICE
Activates or deactivates the Automount function (= automatic connection of the network drive during control start-up). The status of the function is indicated by a check mark under Auto in the network drive table.	AUTO MOUNT

Function	Soft key
Use the ping function to check whether a connection to a particular remote station in the network is available. The address is entered as four decimal numbers separated by points (dotted-decimal notation).	PING
The TNC displays an overview window with information on the active network connections.	NETWORK INFO
Configures access to network drives. (Selectable only after entry of the MOD code number NET123.)	DEFINE NETWORK CONNECTN.
Opens the dialog window for editing the data of an existing network connection. (Selectable only after entry of the MOD code number NET123.)	EDIT NETWORK CONNECTN.
Configures the network address of the control. (Selectable only after entry of the MOD code number NET123.)	CONFIGURE NETWORK
Deletes an existing network connection. (Selectable only after entry of the MOD code number NET123.)	DELETE NETWORK CONNECTN.

Configuring the control's network address

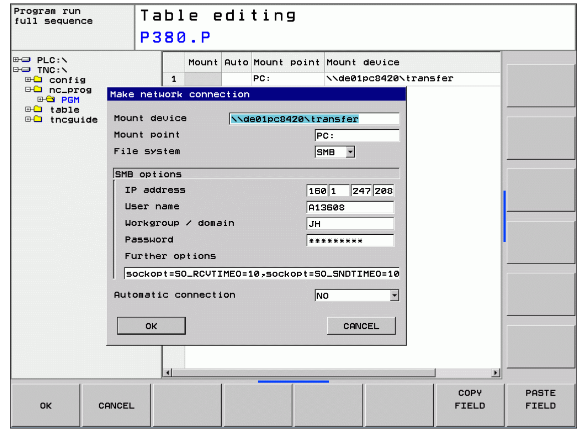
- Connect the TNC (port X26) with a network or a PC
- In the file manager (PGM MGT), select the **Network** soft key
- Press the MOD key. Then enter the keyword **NET123**
- Press the **CONFIGURE NETWORK** soft key to enter the network setting for a specific device (see figure at center right)
- It opens the dialog window for the network configuration



Setting	Meaning
HOSTNAME	Name under which the control logs onto the network. If you use a host-name server, you must enter the "Fully Qualified Host Name" here. If you do not enter a name here, the control uses the so-called null authentication.
DHCP	DHCP = Dynamic Host Configuration Protocol In the drop-down menu, select **YES**, then the control automatically obtains its network address (IP address), the subnet mask, the default router and any required broadcast address from a DHCP server in the network. The DHCP server identifies the control by its hostname. Your company network must be specially prepared for this function. Contact your network administrator.
IP ADDRESS	Network address of the control: In each of the four adjacent input fields you can enter 3 digits of the IP address. With the ENT key you can jump into the next field. Your network specialist can give you a network address for the control.
SUBNET MASK	Serves to distinguish the network ID and the host ID of the network: Your network specialist assigns the subnet mask of the control.
BROADCAST	The broadcast address of the control is needed only if it is different from the standard setting. The standard setting is formed from the net and host ID, in which all bits are set to 1.
ROUTER	Network address of default router: This entry is required only if your network consists of several subnetworks interconnected by routers.

The entered network configuration does not become effective until the control is rebooted. After the network configuration is concluded with the OK button or soft key, the control asks for confirmation and reboots.

Configuring network access to other devices (mount)



> Make sure that the person configuring your TNC is a network specialist.
>
> Not all Windows operating systems require entry of the **username, workgroup** and **password** parameters.

- Connect the TNC (port X26) with a network or a PC
- In the file manager (PGM MGT), select the **Network** soft key
- Press the MOD key. Then enter the keyword **NET123**
- Press the **DEFINE NETWORK CONNECTN.** soft key
- It opens the dialog window for the network configuration

Setting	Meaning
Mount device	■ Connection over NFS: Directory name to be mounted. This is formed from the network address of the device, a colon, a slash and the name of the directory. Entry of the network address as four decimal numbers separated by points (dotted-decimal notation), e.g. 160.1.180.4:/PC. When entering the path name, pay attention to capitalization. ■ To connect individual Windows computers via SMB: Enter the network name and the share name of the computer, e.g. \\PC1791NT\PC
Mount point	Device name: The device name entered here is displayed on the control in the program management for the mounted network, e.g. WORLD: (The name must end with a colon!)
File system	File system type: ■ NFS: Network File System ■ SMB: Windows network
NFS option	**rsize:** Packet size in bytes for data reception **wsize:** Packet size for data transmission in bytes **time0=:** Time in tenths of a second, after which the control repeats an unanswered Remote Procedure Call. **soft:** If **YES** is entered, the Remote Procedure Call is repeated until the NFS server answers. If **NO** is entered, it is not repeated.

Setting	Meaning
SMB option	Options that concern the SMB file system type: Options are given without space characters, separated only by commas. Pay attention to capitalization. Options: **`ip:`** IP address of the Windows PC to which the control is to be connected. **`username:`** User name with which the control should log in. **`workgroup:`** Workgroup under which the control should log in. **`password:`** Password with which the control is to log on (up to 80 characters) Further SMB options: Input of further options for the Windows network
Automatic connection	Automount (YES or NO): Here you specify whether the network will be automatically mounted when the control starts up. Devices that are not automatically mounted can be mounted anytime in the program management.

You do not need to indicate the protocol with the TNC 320. It uses the communications protocol according to RFC 894.

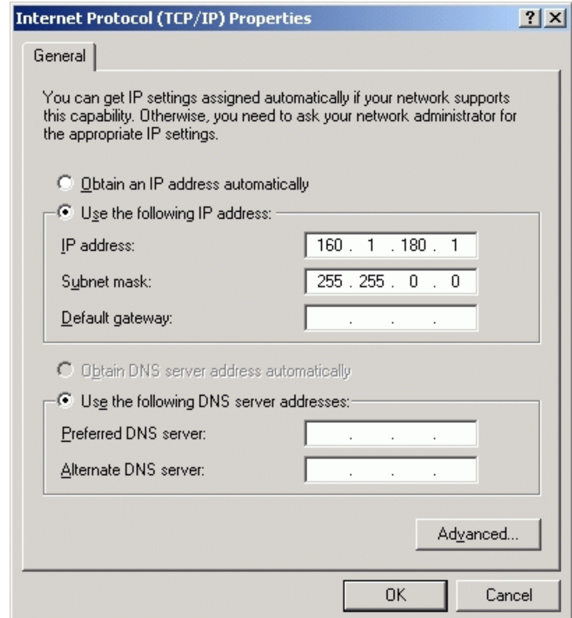
Settings on a PC with Windows 2000

Prerequisite:

The network card must already be installed on the PC and ready for operation.

If the PC that you want to connect the TNC to is already integrated in your company network, then keep the PC's network address and adapt the TNC's network address accordingly.

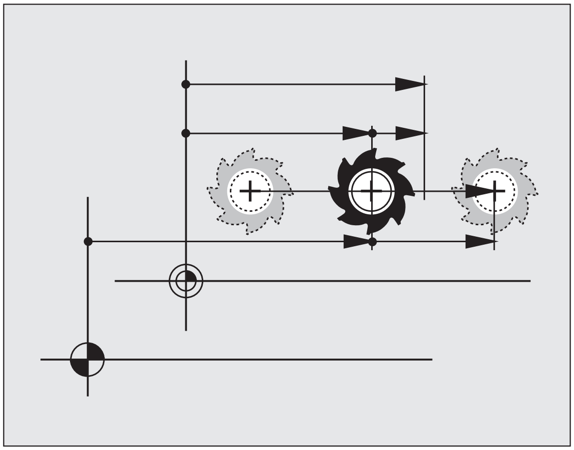
- To open Network Connections, click <Start>, <Control Panel>, <Network and Dial-up Connections>, and then Network Connections
- Right-click the <LAN connection> symbol, and then <Properties> in the menu that appears
- Double-click <Internet Protocol (TCP/IP)> to change the IP settings (see figure at top right)
- If it is not yet active, select the <Use the following IP address> option
- In the <IP address> input field, enter the same IP address that you entered for the PC network settings on the iTNC, e.g. 160.1.180.1
- Enter 255.255.0.0 in the <Subnet mask> input field
- Confirm the settings with <OK>
- Save the network configuration with <OK>. You may have to restart Windows now



15.6 Position Display Types

Application

In the Manual Operation mode and in the Program Run modes of operation, you can select the type of coordinates to be displayed.



The figure at right shows the different tool positions:

- Starting position
- Target position of the tool
- Workpiece datum
- Machine datum

The TNC position displays can show the following coordinates:

Function	Display
Nominal position: the value presently commanded by the TNC	NOML.
Actual position; current tool position	ACTL.
Reference position; the actual position relative to the machine datum	REF ACTL
Reference position; the nominal position relative to the machine datum	REF NOML
Servo lag; difference between nominal and actual positions (following error)	LAG
Distance remaining to the programmed position; difference between actual and target positions	DIST

With the MOD function **Position display 1**, you can select the position display in the status display.

With the MOD function **Position display 2**, you can select the position display in the status display.

15.7 Unit of Measurement

Application

This MOD function determines whether the coordinates are displayed in millimeters (metric system) or inches.

- To select the metric system (e.g. X = 15.789 mm), set the Change mm/inches function to mm. The value is displayed to 3 decimal places.
- To select the inch system (e.g. X = 0.6216 inches), set the Change mm/inches function to inches. The value is displayed to 4 decimal places.

If you would like to activate the inch display, the TNC shows the feed rate in inch/min. In an inch program you must enter the feed rate larger by a factor of 10.

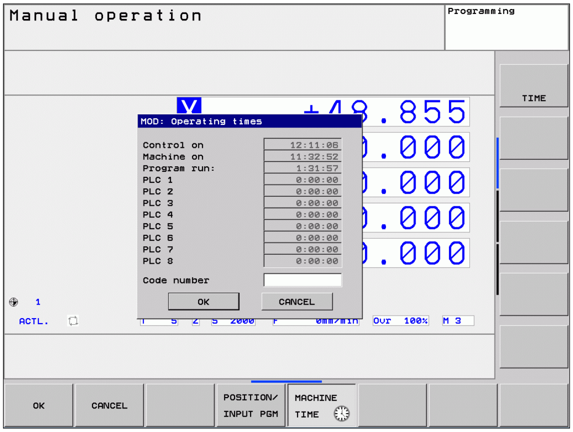
15.8 Displaying Operating Times

Application

The MACHINE TIME soft key enables you to see various types of operating times:

Operating time	Meaning
`Control on`	Operating time of the control since being put into service
`Machine on`	Operating time of the machine tool since being put into service
`Program run`	Duration of controlled operation since being put into service

The machine tool builder can provide further operating time displays. The machine tool manual provides further information.



16

Tables and Overviews

16.1 Machine-Specific User Parameters

Application

The parameter values are entered in the **configuration editor**.

To enable you to set machine-specific functions, your machine tool builder can define which machine parameters are available as user parameters. Furthermore, your machine tool builder can integrate additional machine parameters, which are not described in the following, into the TNC.

Refer to your machine manual.

The machine parameters are grouped as parameter objects in a tree structure in the configuration editor. Each parameter object has a name (e.g. **CfgDisplayLanguage**) that gives information about the parameters it contains. A parameter object, also called "entity", is marked with an "E" in the folder symbol in the tree structure. Some machine parameters have a key name to identify them unambiguously. The key name assigns the parameter to a group (e.g. X for X axis). The respective group folder bears the key name and is marked by a "K" in the folder symbol.

If you are in the configuration editor for the user parameters, you can change the display of the existing parameters. In the default setting, the parameters are displayed with short, explanatory texts. To display the actual system names of the parameters, press the key for the screen layout and then the SHOW SYSTEM NAME soft key. Follow the same procedure to return to the standard display.

Calling the configuration editor

- Select the **`Programming`** mode of operation
- Press the **MOD** key
- Enter the code number **123**
- Press the **`END`** soft key to exit the configuration editor

The icon at the beginning of each line in the parameter tree shows additional information about this line. The icons have the following meanings:

- Branch exists but is closed
- Branch is open
- Empty object, cannot be opened
- Initialized machine parameter
- Uninitialized (optional) machine parameter
- Can be read but not edited
- Cannot be read or edited

The type of the configuration object is identified by its folder symbol:

- Key (group name)
- List
- Entity or parameter object

Displaying help texts

The **HELP** key enables you to call a help text for each parameter object or attribute.

If the help text does not fit on one page (1/2 is then displayed at the upper right, for example), press the **HELP PAGE** soft key to scroll to the second page.

To exit the help text, press the **HELP** key again.

Additional information, such as the unit of measure, the initial value, or a selection list, is also displayed. If the selected machine parameter matches a parameter in the TNC, the corresponding MP number is shown.

Parameter list

Parameter Settings

DisplaySettings

Settings for screen display
Sequence of the displayed axes
[0] to [5]
Depends on the available axes
Type of position display in the position window
NOML.
ACTL.
REF ACTL
REF NOML
LAG
DIST
Type of position display in the status display:
NOML.
ACTL.
REF ACTL
REF NOML
LAG
DIST

Definition of decimal separator for position display

.
Feed rate display in Manual Operation operating mode
At axis key: Display feed rate only if axis-direction key is pressed
Always minimum: Always display feed rate
Display of spindle position in the position display
During closed loop: Display spindle position only if spindle is in position control loop
During closed loop and M5: Display spindle position only if spindle is in position control loop and with M5
hidePresetTable
True: Soft key preset table is not displayed
False: Display soft key preset table

Parameter Settings
DisplaySettings Display step for the individual axes List of all available axes Display step for position display in mm or degrees **0.1** **0.05** **0.01** **0.005** **0.001** **0.0005** **0.0001** **0.00005 (Display step software option)** **0.00001 (Display step software option)** Display step for position display in inches **0.005** **0.001** **0.0005** **0.0001** **0.00005 (Display step software option)** **0.00001 (Display step software option)**
DisplaySettings Definition of the unit of measure valid for the display **Metric: Use metric system** **Inch: Use inch system**
DisplaySettings Format of the NC programs and cycle display Program entry in HEIDENHAIN plain language or in DIN/ISO **HEIDENHAIN: Program entry in plain language in MDI mode** **ISO: Program entry in MDI mode in DIN/ISO format** Display of cycles **TNC_STD: Display cycles with comments** **TNC_PARAM: Display cycles without comments**

Parameter Settings
DisplaySettings Settings of the NC and PLC conversational language NC conversational language **ENGLISH** **GERMAN** **CZECH** **FRENCH** **ITALIAN** **SPANISH** **PORTUGUESE** **SWEDISH** **DANISH** **FINNISH** **DUTCH** **POLISH** **HUNGARIAN** **RUSSIAN** **CHINESE** **CHINESE_TRAD** **SLOVENIAN** **ESTONIAN** **KOREAN** **LATVIAN** **NORWEGIAN** **ROMANIAN** **SLOVAK** **TURKISH** **LITHUANIAN** PLC conversational language **See NC conversational language** Language for PLC error messages **See NC conversational language** Language for online help **See NC conversational language**
DisplaySettings Behavior during control startup Acknowledge the "Power interrupted" message **TRUE: Start-up of the control is not continued until the message has been acknowledged** **FALSE: The "Power interrupted" message does not appear** Display of cycles **TNC_STD: Display cycles with comments** **TNC_PARAM: Display cycles without comments**

Parameter Settings
ProbeSettings
Configuration of probing behavior Manual operation: Including basic rotation **TRUE: Including active basic rotation during probing** **FALSE: Always move on paraxial path during probing** Automatic mode: Multiple measurements in probing functions **1 to 3: Probe points per probing process** Automatic mode: Confidence interval of multiple measurements **0.002 to 0.999 [mm]: Range within which the measured value must be during multiple measurements**
CfgTTRoundStylus Coordinates of the stylus center **[0]: X coordinate of the stylus center with respect to the machine datum** **[1]: Y coordinate of the stylus center with respect to the machine datum** **[2]: Z coordinate of the stylus center with respect to the machine datum** Safety clearance above the stylus for pre-positioning **0.001 to 99 999.9999 [mm]: Set-up clearance in tool-axis direction** Safety zone around the stylus for pre-positioning **0.001 to 99 999.9999 [mm]: Set-up clearance in the plane perpendicular to the tool axis**
CfgToolMeasurement M function for spindle orientation **-1: Spindle orientation directly by the NC** **0: Function inactive** **1 to 999: Number of the M function for spindle orientation** Probing direction for tool radius measurement **X_Positive, Y_Positive, X_Negative, Y_Negative (depending on the tool axis)** Distance from lower edge of tool to upper edge of stylus **0.001 to 99.9999 [mm]: Offset of stylus to tool** Rapid traverse in probing cycle **10 to 300 000 [mm/min]: Rapid traverse in probing cycle** Probing feed rate for tool measurement **1 to 3 000 [mm/min]: Rapid traverse during tool measurement** Calculation of the probing feed rate **ConstantTolerance: Calculation of the probing feed rate with constant tolerance** **VariableTolerance: Calculation of the probing feed rate with variable tolerance** **ConstantFeed: Constant probing feed rate** Max. permissible surface cutting speed at the tooth edge **1 to129 [m/min]: Permissible surface cutting speed at the circumference of the milling tool** Maximum permissible speed during tool measurement **0 to 1 000 [1/min]: Maximum permissible speed** Maximum permissible measuring error for tool measurement **0.001 to 0.999 [mm]: First maximum permissible measurement error** Maximum permissible measuring error for tool measurement **0.001 to 0.999 [mm]: Second maximum permissible measurement error**

Parameter Settings

ChannelSettings
  CH_NC
    Active kinematics
      Kinematics to be activated
        List of machine kinematics
 	
    Geometry tolerances
      Permissible deviation from the radius
        0.0001 to 0.016 [mm]: Permissible deviation of the radius at the circle end-point compared with the circle start-point
 	
    Configuration of the fixed cycles
      Overlap factor for pocket milling
        0.001 to 1.414: Overlap factor for Cycle 4 POCKET MILLING and Cycle 5 CIRCULAR POCKET MILLING
      Display the "Spindle?" error message if M3/M4 is not active
        On: Issue error message
        Off: No error message
      Display the "Enter a negative depth" error message
        On: Issue error message
        Off: No error message
      Behavior when moving to wall of slot in the cylinder surface
        LineNormal. Approach on a straight line
        CircleTangential: Approach on a circular path
      M function for spindle orientation
        -1: Spindle orientation directly by the NC
        0: Function inactive
        1 to 999: Number of the M function for spindle orientation

Parameter Settings

Geometry filter for culling linear elements

Type of stretch filter
- Off: No filter active
- ShortCut: Omit individual points on a polygon
- Average: The geometry filter smooths corners

Maximum distance of the filtered to the unfiltered contour
0 to 10 [mm]: The filtered points lie within this tolerance to the resulting new path

Maximum length of the path as a result of filtering
0 to 1000 [mm]: Length over which geometry filtering is active

Settings for the NC editor

Generate backup files
TRUE: Generate backup file after editing NC programs
FALSE: Do not generate backup file after editing NC programs

Behavior of the cursor after deletion of lines
TRUE: Cursor is placed on the preceding line after deletion (iTNC behavior)
FALSE: Cursor is placed on the following line after deletion

Behavior of the cursor on the first or last line
TRUE: Cursor jumps from end to beginning of program
FALSE: Cursor does not jump from end to beginning of program

Line break with multiline blocks
ALL: Always display all lines
ACT: Only display the lines of the active block completely
NO: Only display all lines when block is edited

Activate help
TRUE: Always display help graphics during input
FALSE: Only display help graphics if HELP was activated by pressing the key

Behavior of the soft-key row after a cycle entry
TRUE: The cycle soft-key row remains active after a cycle definition
FALSE: The cycle soft-key row is hidden after a cycle definition

Safety check when deleting blocks
TRUE: Display confirmation question when deleting an NC block
FALSE: Do not display confirmation question when deleting an NC block

Program length for which the geometry is to be checked
100 to 9999: Program length for which the geometry is to be checked

---

Paths for the end user

List of drives and/or directories
Drives or directories entered here are shown in the TNC's file manager

---

Universal Time (Greenwich Mean Time)

Time difference to universal time [h]
-12 to 13: Time difference in hours relative to Greenwich Mean Time

---

16.2 Pin Layouts and Connecting Cables for the Data Interfaces

RS-232-C/V.24 interface for HEIDENHAIN devices

The interface complies with the requirements of EN 50 178 for **low voltage electrical separation**.

When using the 25-pin adapter block:

TNC		Connecting cable 365 725-xx			Adapter block 310 085-01		Connecting cable 274 545-xx		
Male	**Assignment**	**Female**	**Color**	**Female**	**Male**	**Female**	**Male**	**Color**	**Female**
1	Do not assign	1		1	1	1	1	White/Brown	1
2	RXD	2	Yellow	3	3	3	3	Yellow	2
3	TXD	3	Green	2	2	2	2	Green	3
4	DTR	4	Brown	20	20	20	20	Brown	8
5	Signal GND	5	Red	7	7	7	7	Red	7
6	DSR	6	Blue	6	6	6	6		6
7	RTS	7	Gray	4	4	4	4	Gray	5
8	CTR	8	Pink	5	5	5	5	Pink	4
9	Do not assign	9					8	Violet	20
Hsg.	Ext. shield	Hsg.	Ext. shield	Hsg.	Hsg.	Hsg.	Hsg.	Ext. shield	Hsg.

When using the 9-pin adapter block:

TNC		Connecting cable 355 484-xx			Adapter block 363 987-02		Connecting cable 366 964-xx		
Male	**Assignment**	**Female**	**Color**	**Male**	**Female**	**Male**	**Female**	**Color**	**Female**
1	Do not assign	1	Red	1	1	1	1	Red	1
2	RXD	2	Yellow	2	2	2	2	Yellow	3
3	TXD	3	White	3	3	3	3	White	2
4	DTR	4	Brown	4	4	4	4	Brown	6
5	Signal GND	5	Black	5	5	5	5	Black	5
6	DSR	6	Violet	6	6	6	6	Violet	4
7	RTS	7	Gray	7	7	7	7	Gray	8
8	CTR	8	White/Green	8	8	8	8	White/Green	7
9	Do not assign	9	Green	9	9	9	9	Green	9
Hsg.	Ext. shield	Hsg.	Ext. shield	Hsg.	Hsg.	Hsg.	Hsg.	Ext. shield	Hsg.

Non-HEIDENHAIN devices

The connector layout of a non-HEIDENHAIN device may substantially differ from that of a HEIDENHAIN device.

It depends on the unit and the type of data transfer. The table below shows the connector pin layout on the adapter block.

Adapter block 363 987-02		Connecting cable 366 964-xx		
Female	**Male**	**Female**	**Color**	**Female**
1	1	1	Red	1
2	2	2	Yellow	3
3	3	3	White	2
4	4	4	Brown	6
5	5	5	Black	5
6	6	6	Violet	4
7	7	7	Gray	8
8	8	8	White/Green	7
9	9	9	Green	9
Hsg.	Hsg.	Hsg.	External shield	Hsg.

Ethernet interface RJ45 socket

Maximum cable length:

- Unshielded: 100 m
- Shielded: 400 m

Pin	Signal	Description
1	TX+	Transmit Data
2	TX–	Transmit Data
3	REC+	Receive Data
4	Vacant	
5	Vacant	
6	REC–	Receive Data
7	Vacant	
8	Vacant	

16.3 Technical Information

Explanation of symbols

■ Standard
□ Axis option
◆ Software option 1s

User functions	
Short description	■ Basic version: 3 axes plus closed-loop spindle □ 1st additional axis for 4 axes plus closed-loop spindle □ 2nd additional axis for 5 axes plus closed-loop spindle
Program entry	In HEIDENHAIN conversational format and DIN/ISO over soft keys or USB keyboard
Position data	■ Nominal positions for lines and arcs in Cartesian coordinates or polar coordinates ■ Incremental or absolute dimensions ■ Display and entry in mm or inches
Tool compensation	■ Tool radius in the working plane and tool length ■ Radius compensated contour look ahead for up to 99 blocks (M120)
Tool tables	Multiple tool tables with any number of tools
Constant cutting speed	■ With respect to the path of the tool center ■ With respect to the cutting edge
Parallel operation	Creating a program with graphical support while another program is being run
Contour elements	■ Straight line ■ Chamfer ■ Circular path ■ Circle center point ■ Circle radius ■ Tangentially connected arc ■ Corner rounding
Approaching and departing the contour	■ Via straight line: tangential or perpendicular ■ Via circular arc
FK free contour programming	■ FK free contour programming in HEIDENHAIN conversational format with graphic support for workpiece drawings not dimensioned for NC
Program jumps	■ Subroutines ■ Program-section repeat ■ Any desired program as subroutine

User functions	
Fixed cycles	■ Cycles for drilling, and conventional and rigid tapping ■ Roughing of rectangular and circular pockets ■ Cycles for pecking, reaming, boring, and counterboring ■ Cycles for milling internal and external threads ■ Finishing of rectangular and circular pockets ■ Cycles for clearing level and inclined surfaces ■ Cycles for milling linear and circular slots ■ Linear and circular point patterns ■ Contour-parallel contour pocket ■ Contour train ■ OEM cycles (special cycles developed by the machine tool builder) can also be integrated
Coordinate transformation	■ Datum shift, rotation, mirroring ■ Scaling factor (axis-specific) ◆ Tilting the working plane (software option)
Q parameters Programming with variables	■ Mathematical functions =, +, –, *, /, sin α, cos α, root calculation ■ Logical comparisons (=, =/ , <, >) ■ Calculating with parentheses ■ tan α, arc sin, arc cos, arc tan, a^n, e^n, ln, log, absolute value of a number, the constant π, negation, truncation of digits before or after the decimal point ■ Functions for calculation of circles ■ String parameters
Programming aids	■ Online calculator ■ Complete list of all current error messages ■ Context-sensitive help function for error messages ■ Graphic support for the programming of cycles ■ Comment blocks in the NC program
Actual position capture	■ Actual positions can be transferred directly into the NC program
Test run graphics Display modes	■ Graphic simulation before program run, even while another program is being run ■ Plan view / projection in 3 planes / 3-D view ■ Magnification of details
Programming graphics	■ In the Programming mode, the contour of the NC blocks is drawn on screen while they are being entered (2-D pencil-trace graphics), even while another program is running
Program Run graphics Display modes	■ Graphic simulation of real-time machining in plan view / projection in 3 planes / 3-D view
Machining time	■ Calculating the machining time in the Test Run mode of operation ■ Display of the current machining time in the Program Run modes
Returning to the contour	■ Mid-program startup in any block in the program, returning the tool to the calculated nominal position to continue machining ■ Program interruption, contour departure and return

User functions	
Datum tables	■ Multiple datum tables, for storing workpiece-related datums
Touch-probe cycles	■ Touch probe calibration ■ Compensation of workpiece misalignment, manual or automatic ■ Datum setting, manual or automatic ■ Automatic workpiece measurement ■ Cycles for automatic tool measurement

Specifications	
Components	■ Main computer with TNC keyboard and integrated 15.1-inch TFT color flat-panel display with soft keys
Program memory	■ 300 MB (on CFR compact flash memory card)
Input resolution and display step	■ Up to 0.1 µm for linear axes ■ Up to 0.0001° for angular axes
Input range	■ Maximum 999 999 999 mm or 999 999 999°
Interpolation	■ Linear in 4 axes ■ Circular in 2 axes ◆ Circular in 3 axes with tilted working plane (software option 1) ■ Helical: superimposition of circular and straight paths
Block processing time 3-D straight line without radius compensation	■ 6 ms (3-D straight line without radius compensation)
Axis feedback control	■ Position loop resolution: Signal period of the position encoder/1024 ■ Cycle time of position controller: 3 ms ■ Cycle time of speed controller: 600 µs
Range of traverse	■ Maximum 100 m (3937 inches)
Spindle speed	■ Maximum 100 000 rpm (analog speed command signal)
Error compensation	■ Linear and nonlinear axis error, backlash, reversal peaks during circular movements, thermal expansion ■ Static friction
Data interfaces	■ One each RS-232-C /V.24 max. 115 kilobaud ■ Expanded data interface with LSV-2 protocol for remote operation of the TNC through the data interface with the HEIDENHAIN software TNCremo ■ Ethernet interface 100BaseT Approx. 2 to 5 megabaud (depending on file type and network load) ■ 3 x USB 2.0
Ambient temperature	■ Operation: 0 °C to +45 °C ■ Storage: –30 °C to +70 °C

Accessories	
Electronic handwheels	■ One **HR 410** portable handwheel, or ■ One **HR 130** panel-mounted handwheel, or ■ Up to three **HR 150** panel-mounted handwheels via HRA 110 handwheel adapter
Touch probes	■ **TS 220**: 3-D touch trigger probe with cable connection, or ■ **TS 440**: 3-D touch trigger probe with infrared transmission ■ **TS 444**: Battery-free 3-D touch trigger probe with infrared transmission ■ **TS 640**: 3-D touch trigger probe with infrared transmission ■ **TS 740**: High-precision 3-D touch trigger probe with infrared transmission ■ **TT 140**: 3-D touch trigger probe for tool measurement

Software option 1 (option number #08)	
Rotary table machining	◆ Programming of cylindrical contours as if in two axes ◆ Feed rate in distance per minute
Coordinate transformation	◆ Tilting the working plane
Interpolation	◆ Circle in 3 axes with tilted working plane

Input format and unit of TNC functions	
Positions, coordinates, circle radii, chamfer lengths	–99 999.9999 to +99 999.9999 (5, 4: places before the decimal point, places after the decimal point) [mm]
Tool numbers	0 to 32 767.9 (5.1)
Tool names	16 characters, enclosed by quotation marks with `TOOL CALL`. Permitted special characters: #, $, %, &, -
Delta values for tool compensation	–99.9999 to +99.9999 (2.4) [mm]
Spindle speeds	0 to 99 999.999 (5.3) [rpm]
Feed rates	0 to 99 999.999 (5.3) [mm/min] or [mm/tooth] or [mm/rev]
Dwell time in Cycle 9	0 to 3600.000 (4.3) [s]
Thread pitch in various cycles	–99.9999 to +99.9999 (2.4) [mm]
Angle of spindle orientation	0 to 360.0000 (3.4) [°]
Angle for polar coordinates, rotation, tilting the working plane	–360.0000 to +360.0000 (3.4) [°]
Polar coordinate angle for helical interpolation (CP)	-5 400.0000 to 5 400.0000 (4.4) [°]
Datum numbers in Cycle 7	0 to 2999 (4.0)
Scaling factor in Cycles 11 and 26	0.000 001 to 99.999 999 (2.6)
Miscellaneous functions M	0 to 999 (3.0)
Q parameter numbers	0 to 1999 (4.0)
Q parameter values	-99 999.9999 to +99 999.9999 (5.4)
Surface-normal vectors N and T with 3-D compensation	-9.99999999 to +9.99999999 (1.8)
Labels (LBL) for program jumps	0 to 999 (3.0)
Labels (LBL) for program jumps	Any text string in quotes (“ ”)
Number of program section repeats REP	1 to 65 534 (5.0)
Error number with Q parameter function FN14	0 to 1099 (4.0)

16.4 Exchanging the Buffer Battery

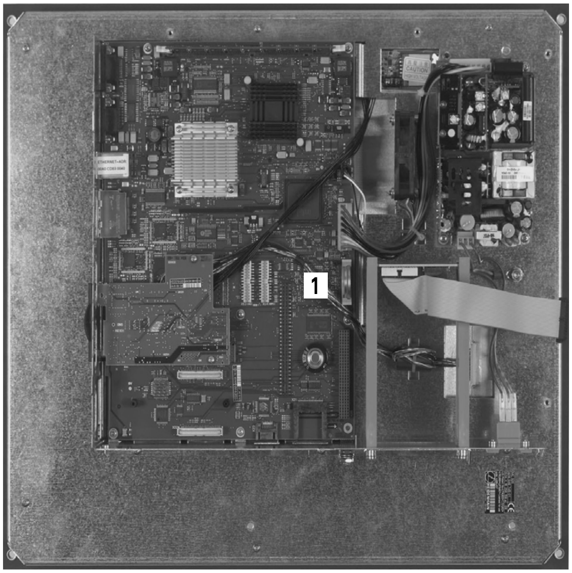
A buffer battery supplies the TNC with current to prevent the data in RAM memory from being lost when the TNC is switched off.

If the TNC displays the error message **Exchange buffer battery**, then you must replace the battery:

Make a data backup before changing the buffer battery!

To exchange the buffer battery, first switch off the TNC!

The buffer battery must be exchanged only by trained service personnel.



Battery type: 1 Lithium battery, type CR 2450N (Renata)
ID 315 878-01

1. The buffer battery is on the main board of the MC 6110
2. Remove the five screws of the MC 6110 housing cover
3. Remove the cover
4. The buffer battery is at the border of the PCB
5. Exchange the battery. The socket accepts a new battery only in the correct polarity

Overview Tables

Fixed cycles

Cycle number	Cycle designation	DEF active	CALL active
7	Datum shift	■	
8	Mirror image	■	
9	Dwell time	■	
10	Rotation	■	
11	Scaling factor	■	
12	Program call	■	
13	Oriented spindle stop	■	
14	Contour definition	■	
19	Tilting the working plane	■	
20	Contour data SL II	■	
21	Pilot drilling SL II		■
22	Rough out SL II		■
23	Floor finishing SL II		■
24	Side finishing SL II		■
25	Contour train		■
26	Axis-specific scaling	■	
27	Cylinder surface		■
28	Cylindrical surface slot		■
29	Cylinder surface ridge		■
32	Tolerance	■	
200	Drilling		■
201	Reaming		■
202	Boring		■
203	Universal drilling		■
204	Back boring		■
205	Universal pecking		■

Cycle number	Cycle designation	DEF active	CALL active
206	Tapping with a floating tap holder, new		■
207	Rigid tapping, new		■
208	Bore milling		■
209	Tapping with chip breaking		■
220	Circular point pattern	■	
221	Linear point pattern	■	
230	Multipass milling		■
231	Ruled surface		■
232	Face milling		■
240	Centering		■
241	Single-lip deep-hole drilling		■
247	Datum setting	■	
251	Rectangular pocket (complete machining)		■
252	Circular pocket (complete machining)		■
253	Slot milling		■
254	Circular slot		■
256	Rectangular stud (complete machining)		■
257	Circular stud (complete machining)		■
262	Thread milling		■
263	Thread milling/countersinking		■
264	Thread drilling/milling		■
265	Helical thread drilling/milling		■
267	Outside thread milling		■

Miscellaneous functions

M	Effect	Effective at block... Start	End	Page
M0	Program run STOP/Spindle STOP/Coolant OFF		■	Page 271
M1	Optional program STOP/Spindle STOP/Coolant OFF		■	Page 392
M2	Program run STOP/Spindle STOP/Coolant OFF/Clear status display (depending on machine parameter)/Go to block 1		■	Page 271
M3 M4 M5	Spindle ON clockwise Spindle ON counterclockwise Spindle STOP	■ ■ 	 ■	Page 271
M6	Tool change/Stop program run (depending on machine parameter)/Spindle STOP		■	Page 271
M8 M9	Coolant ON Coolant OFF	■ 	 ■	Page 271
M13 M14	Spindle ON clockwise/Coolant ON Spindle ON counterclockwise/Coolant ON	■ ■		Page 271
M30	Same function as M2		■	Page 271
M89	Vacant miscellaneous function **or** Cycle call, modally effective (depending on machine parameter)	■ 	 ■	Cycles Manual
M91	Within the positioning block: Coordinates are referenced to machine datum	■		Page 272
M92	Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position	■		Page 272
M94	Reduce the rotary axis display to a value below 360°	■		Page 322
M97	Machine small contour steps		■	Page 275
M98	Machine open contours completely		■	Page 277
M99	Blockwise cycle call		■	Cycles Manual

M	Effect	Effective at block... Start	End	Page
M101	Automatic tool change with replacement tool if maximum tool life has expired		■	Page 146
M102	Reset M101		■	
M109	Constant contouring speed at tool cutting edge (increase and decrease feed rate)	■		Page 279
M110	Constant contouring speed at tool cutting edge (feed rate decrease only)	■		
M111	Reset M109/M110		■	
M116	Feed rate for rotary axes in mm/min	■		Page 320
M117	Reset M116		■	
M118	Superimpose handwheel positioning during program run	■		Page 282
M120	Pre-calculate radius-compensated contour (LOOK AHEAD)	■		Page 280
M126	Shortest-path traverse of rotary axes	■		Page 321
M127	Reset M126		■	
M130	Moving to position in an untilted coordinate system with a tilted working plane	■		Page 274
M140	Retraction from the contour in the tool-axis direction	■		Page 283
M144	Compensating the machine’s kinematics configuration for ACTUAL/NOMINAL positions at end of block	■		Page 324
M145	Reset M144		■	
M141	Suppress touch probe monitoring	■		Page 284
M148	Retract the tool automatically from the contour at NC stop	■		Page 285
M149	Reset M148		■	

Comparison: Functions of the TNC 320 and the iTNC 530

Comparison: Specifications

Function	TNC 320	iTNC 530
Axes	5 maximum	18 maximum
Input resolution and display step:		
■ Linear axes	■ 1 µm	■ 0.1 µm
■ Rotary axes	■ 0.001°	■ 0.0001°
Display	15.1-inch TFT color flat-panel display	15.1-inch TFT color flat-panel display (optional: 19-inch TFT)
Memory media for NC, PLC programs and system files	CompactFlash memory card	Hard disk
Program memory for NC programs	300 MB	25 GB
Block processing time	6 ms	3.6 ms (MC 420) 0.5 ms (MC 422 C)
HeROS operating system	Yes	Yes
Windows XP operating system	No	Option
Interpolation:		
■ Straight line	■ 4 axes	■ 5 axes
■ Circle	■ 3 axes	■ 3 axes
■ Helix	■ Yes	■ Yes
■ Spline	■ No	■ Yes, option with MC 420
Hardware	Compact in operating panel	Modular in electrical cabinet

Comparison: Data interfaces

Function	TNC 320	iTNC 530
100BaseT Fast Ethernet	X	X
RS-232-C/V.24 serial interface	X	X
RS-422/V.11 serial interface	-	X
USB interface	X (USB 2.0)	X (USB 1.1)

Comparison: Accessories

Function	TNC 320	iTNC 530
Machine operating panel		
■ MB 420	■ –	■ X
■ MB 620 (HSCI)	■ X	■ X
Electronic handwheels		
■ HR 410	■ X	■ X
■ HR 420	■ –	■ X
■ HR 520/530/550	■ –	■ X
■ HR 130	■ X	■ X
■ HR 150 via HRA 110	■ X	■ X
Touch probes		
■ TS 220	■ X	■ X
■ TS 440	■ X	■ X
■ TS 444	■ X	■ X
■ TS 449 / TT 449	■ –	■ X
■ TS 640	■ X	■ X
■ TS 740	■ X	■ X
■ TT 130 / TT 140	■ X	■ X
Industrial PC **IPC 61xx**	–	X

Comparison: PC software

Function	TNC 320	iTNC 530
Programming station software	Available	Available
TNCremoNT for data transfer with **TNCbackup** for data backup	Available	Available
TNCremoPlus data transfer software with "live" screen	Available	Available
RemoTools SDK 1.2: Function library for developing your own applications for communicating with HEIDENHAIN controls	Limited functionality available	Available
virtualTNC: Control component for virtual machines	Not available	Available
ConfigDesign: Software for configuring the control	Available	Not available

Comparison: Machine-specific functions

Function	TNC 320	iTNC 530
Switching the traverse range	Function not available	Available function
Central drive (1 motor for multiple machine axes)	Function not available	Available function
C-axis operation (spindle motor drives rotary axis)	Function not available	Available function
Automatic exchange of milling head	Function not available	Available function
Support of angle heads	Function not available	Available function
Balluf tool identification	Function not available	Available function
Management of multiple tool magazines	Function not available	Available function
Expanded tool management via Python	Function not available	Available function

Comparison: User functions

Function	TNC 320	iTNC 530
Program entry		
■ HEIDENHAIN conversational	■ X	■ X
■ DIN/ISO	■ X (soft keys)	■ X (ASCII keys)
■ With smarT.NC	■ –	■ X
■ With ASCII editor	■ X, directly editable	■ X, editable after conversion
Position data		
■ Nominal positions for lines and arcs in Cartesian coordinates	■ X	■ X
■ Nominal positions for lines and arcs in polar coordinates	■ X	■ X
■ Incremental or absolute dimensions	■ X	■ X
■ Display and entry in mm or inches	■ X	■ X
■ Paraxial positioning blocks	■ X	■ X
■ Set the last tool position as pole (empty CC block)	■ X (error message if pole transfer is ambiguous)	■ X
■ Surface normal vectors (**LN**)	■ –	■ X
■ Spline blocks (**SPL**)	■ –	■ X
Tool compensation		
■ In the working plane, and tool length	■ X	■ X
■ Radius compensated contour look ahead for up to 99 blocks	■ X	■ X
■ Three-dimensional tool radius compensation	■ –	■ X
Tool table		
■ Central storage of tool data	■ X, variable numbering	■ X, fixed numbering
■ Multiple tool tables with any number of tools	■ X	■ X
■ Flexible management of tool types	■ X	■ –
■ Filtered display of selectable tools	■ X	■ –
■ Sorting function	■ X	■ –
■ Column names	■ Sometimes with _	■ Sometimes with **-**
■ Copy function: Overwriting relevant tool data	■ –	■ X
■ Form view	■ Switchover with split-screen layout key	■ Switchover by soft key
■ Exchange of tool table between TNC 320 and iTNC 530	■ Not possible	■ Not possible
Touch-probe table for managing different 3-D touch probes	X	–

Function	TNC 320	iTNC 530
Creating tool-usage file, checking the availability	–	X
Cutting-data tables: Automatic calculation of spindle speed and feed rate from saved technology tables	–	X
Define any tables	■ Definable via config. data ■ Table names must start with a letter ■ Reading and writing with SQL functions	■ Freely definable tables (.TAB files) ■ Reading and writing with FN functions
Constant contouring speed: Relative to the path of the tool center or relative to the tool's cutting edge	X	X
Parallel operation: Creating programs while another program is being run	X	X
Programming of counter axes	–	X
Tilting the working plane (Cycle 19, PLANE function)	Option #08	X, option #08 with MC 420
Machining with rotary tables		
■ Programming of cylindrical contours as if in two axes	■	■
■ Cylinder Surface (Cycle 27)	■ X, option #08	■ X, option #08 with MC 420
■ Cylinder Surface (Cycle 28)	■ X, option #08	■ X, option #08 with MC 420
■ Cylinder Surface Ridge (Cycle 29)	■ X, option #08	■ X, option #08 with MC 420
■ Cylinder Surface External Contour (Cycle 39)	■ –	■ X, option #08 with MC 420
■ Feed rate in mm/min or rev/min	■ X, option #08	■ X, option #08 with MC 420
Traverse in tool-axis direction		
■ Manual operation (3-D ROT menu)	■ –	■ X, FCL2 function
■ During program interruption	■ –	■ X
■ With handwheel superimpositioning	■ –	■ X, option #44
Approaching and departing the contour: Via a straight line or arc	X	X
Entry of feed rates:		
■ **F** (mm/min), rapid traverse **FMAX**	■ X	■ X
■ **FU** (feed per revolution mm/rev)	■ X	■ X
■ **FZ** (tooth feed rate)	■ X	■ X
■ **FT** (time in seconds for path)	■ –	■ X
■ **FMAXT** (only for active rapid traverse pot: time in seconds for path)	■ –	■ X

Function	TNC 320	iTNC 530
FK free contour programming		
■ Programming for workpiece drawings not dimensioned for NC programming	■ X	■ X
■ Conversion of FK program to conversational dialog	■ –	■ X
Program jumps:		
■ Maximum number of label numbers	■ 65535	■ 1000
■ Subroutines	■ X	■ X
■ Nesting depth for subprograms	■ 20	■ 6
■ Program section repeats	■ X	■ X
■ Any desired program as subroutine	■ X	■ X
Q parameter programming:		
■ Standard mathematical functions	■ X	■ X
■ Formula entry	■ X	■ X
■ String processing	■ X	■ X
■ Local Q parameters **QL**	■ –	■ X
■ Nonvolatile Q parameters **QR**	■ –	■ X
■ Changing parameters during program interruption	■ –	■ X
■ **FN15:PRINT**	■ –	■ X
■ **FN25:PRESET**	■ –	■ X
■ **FN26:TABOPEN**	■ –	■ X
■ **FN27:TABWRITE**	■ –	■ X
■ **FN28:TABREAD**	■ –	■ X
■ **FN29: PLC LIST**	■ X	■ –
■ **FN31: RANGE SELECT**	■ –	■ X
■ **FN32: PLC PRESET**	■ –	■ X
■ **FN37:EXPORT**	■ X	■ –
■ **FN38: SEND**	■ –	■ X
■ Saving file externally with **FN16**	■ –	■ X
■ **FN16** formatting: Left-aligned, right-aligned, string lengths	■ –	■ X
■ **FN16**: Standard behavior while writing the file, if not defined with **M_APPEND** or **M_CLOSE**	■ Each time F16 is called, the protocol is overwritten	■ Each time F16 is called, the data is appended to the existing file
■ Writing to LOG file with **FN16**	■ X	■ –
■ Displaying parameter contents in the additional status display	■ X	■ –
■ Displaying parameter contents during programming (Q-INFO)	■ –	■ X
■ **SQL** functions for writing and reading tables	■ X	■ –

Function	TNC 320	iTNC 530
Graphic support		
■ 2-D programming graphics	■ X	■ X
■ Synchronization between block display and graphics	■ –	■ X
■ REDRAW function	■ –	■ X
■ Show grid lines as the background	■ X	■ –
■ 3-D programming graphics	■ –	■ X
■ Test graphics (plan view, projection in 3 planes, 3-D view)	■ X	■ X
■ High-resolution view	■ –	■ X
■ Image data processing	■ Blockwise	■ In continuous jog mode
■ Tool display	■ Only in plan view	■ X
■ Setting the simulation speed	■ –	■ X
■ Coordinates of line intersection for projection in 3 planes	■ –	■ X
■ Expanded zoom functions (mouse operation)	■ –	■ X
■ Displaying frame for workpiece blank	■ X	■ X
■ Displaying the depth value in plan view during mouse-over	■ –	■ X
■ Targeted stop of test run (STOP AT N)	■ –	■ X
■ Consideration of tool change macro	■ –	■ X
■ Program run graphics (plan view, projection in 3 planes, 3-D view)	■ X	■ X
■ High-resolution view	■ –	■ X
■ Saving/opening of simulation results	■ X	■ –
Datum tables: for storing workpiece-related datums	X	X
Preset table: for saving reference points (presets)	X	X
Pallet management		
■ Support of pallet files	■ –	■ X
■ Tool-oriented machining	■ –	■ X
■ Pallet preset table: for managing pallet datums	■ –	■ X
Returning to the contour		
■ With mid-program startup	■ X	■ X
■ After program interruption	■ X	■ X
Autostart function	X	X
Actual position capture: Actual positions can be transferred to the NC program	X	X
Enhanced file management		
■ Creating multiple directories and subdirectories	■ X	■ X
■ Sorting function	■ X	■ X
■ Mouse operation	■ X	■ X
■ Selection of target directory by soft key	■ –	■ X

Function	TNC 320	iTNC 530
Programming aids:		
Help graphics for cycle programming	X, can be switched off via config datum	X
Animated help graphics when **PLANE/PATTERN DEF** function is selected	–	X
Help graphics for **PLANE/PATTERN DEF**	–	X
Context-sensitive help function for error messages	X	X
TNCguide: Browser-based help system	X	X
Context-sensitive call of help system	–	X
Calculator	X (scientific)	X (standard)
Comment blocks in NC program	X (input via screen keyboard)	X (input via ASCII keyboard)
Structure blocks in NC program	X (input via screen keyboard)	X (input via ASCII keyboard)
Structure view in test run	–	X
Structure view for large programs	–	X
Dynamic Collision Monitoring (DCM):		
Collision monitoring in Automatic operation	–	X, option #40
Collision monitoring in Manual operation	–	X, option #40
Graphic depiction of the defined collision objects	–	X, option #40
Collision checking in the Test Run mode	–	X, option #40
Fixture monitoring	–	X, option #40
Tool carrier management	–	X, option #40
CAM support:		
Loading of contours from DXF data	–	X, option #42
Loading of machining positions from DXF data	–	X, option #42
Offline filter for CAM files	–	X
Stretch filter	x	–
MOD functions:		
User parameters	Configuration data	Numerical structure
OEM help files with service functions	–	X
Data medium inspection	–	X
Loading of service packs	–	X
Setting the system time	–	X
Selection of axes for actual position capture	–	X
Definition of traverse range limits	–	X
Restricting external access	–	X
Switching the kinematics	–	X

Function	TNC 320	iTNC 530
Calling fixed cycles:		
■ With **M99** or **M89**	■ X	■ X
■ With **CYCL CALL**	■ X	■ X
■ With **CYCL CALL PAT**	■ X	■ X
■ With **CYC CALL POS**	■ –	■ X
Special functions:		
■ Creating backward programs	■ –	■ X
■ Datum shift with **TRANS DATUM**	■ –	■ X
■ Adaptive Feed Control (AFC)	■ –	■ X, option #45
■ Global definition of cycle parameters: **GLOBAL DEF**	■ –	■ X
■ Pattern definition with **PATTERN DEF**	■ X	■ X
■ Definition and execution of point tables	■ X	■ X
■ Simple contour formula **CONTOUR DEF**	■ X	■ X
Functions for large molds and dies:		
■ Global program settings (GS)	■ –	■ X, option #44
■ Expanded **M128**: **FUNCTION TCPM**	■ –	■ X
Status displays:		
■ Positions, spindle speed, feed rate	■ X	■ X
■ Larger depiction of position display, Manual Operation	■ –	■ X
■ Additional status display, form view	■ X	■ X
■ Display of handwheel traverse when machining with handwheel superimposition	■ –	■ X
■ Display of distance-to-go in a tilted system	■ –	■ X
■ Dynamic display of Q-parameter contents, definable number ranges	■ X	■ –
■ OEM-specific additional status display via Python	■ –	■ X
■ Graphic display of residual run time	■ –	■ X
Individual color settings of user interface	–	X

Comparison: Cycles

Cycle	TNC 320	iTNC 530
1, pecking	X	X
2, tapping	X	X
3, slot milling	X	X
4, pocket milling	X	X
5, circular pocket	X	X
6, rough out (SL I)	–	X
7, datum shift	X	X
8, mirror image	X	X
9, dwell time	X	X
10, rotation	X	X
11, scaling	X	X
12, program call	X	X
13, oriented spindle stop	X	X
14, contour definition	X	X
15, pilot drilling (SL I)	–	X
16, contour milling (SL I)	–	X
17, tapping (controlled spindle)	X	X
18, thread cutting	X	X
19, working plane	X, option #08	X, option #08 with MC 420
20, contour data	X	X
21, pilot drilling	X	X
22, rough-out: ■ Parameter Q401, feed rate factor ■ Parameter Q404, fine roughing strategy	X ■ – ■ –	X ■ X ■ X
23, floor finishing	X	X
24, side finishing	X	X
25, contour train	X	X
26, axis-specific scaling factor	X	X

Cycle	TNC 320	iTNC 530
27, contour surface	Option #08	X, option #08 with MC 420
28, cylinder surface	Option #08	X, option #08 with MC 420
29, cylinder surface ridge	Option #08	X, option #08 with MC 420
30, 3-D data	–	X
32, tolerance	X	X
32, tolerance with HSC mode and TA	–	X, option #09 with MC 420
39, cylinder surface external contour	–	X, option #08 with MC 420
200, drilling	X	X
201, reaming	X	X
202, boring	X	X
203, universal drilling	X	X
204, back boring	X	X
205, universal pecking	X	X
206, tapping with floating tap holder	X	X
207, rigid tapping, new	X	X
208, bore milling	X	X
209, tapping with chip breaking	X	X
210, slot with reciprocating plunge	X	X
211, circular slot	X	X
212, rectangular pocket finishing	X	X
213, rectangular stud finishing	X	X
214, circular pocket finishing	X	X
215, circular stud finishing	X	X
220, circular pattern	X	X
221, linear pattern	X	X
230, multipass milling	X	X
231, ruled surface	X	X

Cycle	TNC 320	iTNC 530
232, face milling	X	X
240, centering	X	X
241, single-lip deep-hole drilling	X	X
247, datum setting	X	X
251, rectangular pocket (complete)	X	X
252, circular pocket (complete)	X	X
253, slot (complete)	X	X
254, circular slot (complete)	X	X
256, rectangular stud (complete)	X	X
257, circular stud (complete)	X	X
262, thread milling	X	X
263, thread milling/countersinking	X	X
264, thread drilling/milling	X	X
265, helical thread drilling/milling	X	X
267, outside thread milling	X	X
270, contour train data for defining the behavior of Cycle 25	–	X

Comparison: Miscellaneous functions

M	Effect	TNC 320	iTNC 530
M00	Program run STOP/Spindle STOP/Coolant OFF	X	X
M01	Optional program STOP	X	X
M02	STOP program run/Spindle STOP/Coolant OFF/CLEAR status display (depending on machine parameter)/Go to block 1	X	X
M03 M04 M05	Spindle ON clockwise Spindle ON counterclockwise Spindle STOP	X	X
M06	Tool change/Program run STOP (machine-dependent function)/Spindle STOP	X	X
M08 M09	Coolant ON Coolant OFF	X	X
M13 M14	Spindle ON clockwise/Coolant ON Spindle ON counterclockwise/Coolant ON	X	X
M30	Same function as M02	X	X
M89	Vacant miscellaneous function **or** Cycle call, modally effective (machine-dependent function)	X	X
M90	Constant contouring speed at corners	–	X
M91	Within the positioning block: Coordinates are referenced to machine datum	X	X
M92	Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position	X	X
M94	Reduce the rotary axis display to a value below 360°	X	X
M97	Machine small contour steps	X	X
M98	Machine open contours completely	X	X
M99	Blockwise cycle call	X	X
M101 M102	Automatic tool change with replacement tool if maximum tool life has expired Reset M101	–	X
M103	Reduce feed rate during plunging to factor F (percentage)	–	X
M104	Reactivate the datum as last defined	–	X
M105 M106	Machining with second k_V factor Machining with first k_V factor	–	X
M107 M108	Suppress error message for replacement tools with oversize Reset M107	X	X

M	Effect	TNC 320	iTNC 530
M109 M110 M111	Constant contouring speed at tool cutting edge (increase and decrease feed rate) Constant contouring speed at tool cutting edge (feed rate decrease only) Reset M109/M110	X	X
M112 M113	Enter contour transition between two contour elements Reset M112	–	X
M114 M115	Automatic compensation of machine geometry when working with tilted axes Reset M114	–	X, option #08 with MC 420
M116 M117	Feed rate for rotary tables in mm/min Reset M116	Option #08	X, option #08 with MC 420
M118	Superimpose handwheel positioning during program run	X	X
M120	Pre-calculate radius-compensated contour (LOOK AHEAD)	X	X
M124	Contour filter	–	X
M126 M127	Shortest-path traverse of rotary axes Reset M126	X	X
M128 M129	Maintain the position of the tool tip when positioning the tilted axes (TCPM) Reset M126	–	X, option #09 with MC 420
M130	Within the positioning block: Points are referenced to the untilted coordinate system	X	X
M134 M135	Exact stop at nontangential contour transitions when positioning with rotary axes Reset M134	–	X
M136 M137	Feed rate F in millimeters per spindle revolution Reset M136	–	X
M138	Selection of tilted axes	–	X
M140	Retraction from the contour in the tool-axis direction	X	X
M141	Suppress touch probe monitoring	X	X
M142	Delete modal program information	–	X
M143	Delete basic rotation	X	X

M	Effect	TNC 320	iTNC 530
M144 M145	Compensating the machine's kinematics configuration for ACTUAL/NOMINAL positions at end of block Reset M144	Option #08	X, option #09 with MC 420
M148 M149	Retract the tool automatically from the contour at NC stop Reset M148	X	X
M150	Suppress limit switch message	–	X
M200-M204	Laser cutting functions	–	X

Comparison: Touch probe cycles in the Manual Operation and El. Handwheel modes

Cycle	TNC 320	iTNC 530
Touch-probe table for managing 3-D touch probes	X	–
Calibrating the effective length	X	X
Calibrating the effective radius	X	X
Measuring a basic rotation using a line	X	X
Setting the datum in any axis	X	X
Setting a corner as datum	X	X
Setting a circle center as datum	X	X
Setting a center line as datum	–	X
Measuring a basic rotation using two holes/cylindrical studs	–	X
Setting the datum using four holes/cylindrical studs	–	X
Setting the circle center using three holes/cylindrical studs	–	X
Support of mechanical touch probes by manually capturing the current position	By soft key	By hard key
Writing measured values in preset table	X	X
Writing measured values in datum tables	X	X

Comparison: Touch probe cycles for automatic workpiece inspection

Cycle	TNC 320	iTNC 530
0, reference plane	X	X
1, polar datum	X	X
2, calibrate TS	–	X
3, measuring	X	X
4, measuring in 3-D	–	X
9, calibrate TS length	–	X
30, calibrate TT	X	X
31, measure tool length	X	X
32, measure tool radius	X	X
33, measure tool length and radius	X	X
400, basic rotation	X	X
401, basic rotation from two holes	X	X
402, basic rotation from two studs	X	X
403, compensate a basic rotation via a rotary axis	X	X
404, setting a basic rotation	X	X
405, compensating workpiece misalignment by rotating the C axis	X	X
408, slot center datum	X	X
409, ridge center datum	X	X
410, datum from inside of rectangle	X	X
411, datum from outside of rectangle	X	X
412, datum from inside of circle	X	X
413, datum from outside of circle	X	X
414, datum at outside corner	X	X
415, datum at inside corner	X	X
416, datum at circle center	X	X
417, datum in touch probe axis	X	X
418, datum at center of 4 holes	X	X

Cycle	TNC 320	iTNC 530
419, datum in one axis	X	X
420, measure angle	X	X
421, measuring a hole	X	X
422, measuring a circle from outside	X	X
423, measuring a rectangle from inside	X	X
424, measuring a rectangle from outside	X	X
425, measuring inside width	X	X
426, measuring a ridge from outside	X	X
427, boring	X	X
430, measuring a bolt hole circle	X	X
431, measuring a plane	X	X
440, measuring an axis shift	–	X
441, fast probing	–	X
450, saving the kinematics	–	X
451, measuring the kinematics	–	X
452, preset compensation	–	X
480, calibrating a TT	X	X
481, measuring/inspecting the tool length	X	X
482, measuring/inspecting the tool radius	X	X
483, measuring/inspecting the tool length and radius	X	X
484, calibrating the infrared TT	–	X

Comparison: Differences in programming

Function	TNC 320	iTNC 530
Input of texts (comments, program names, structure items, network addresses, etc.)	Input via screen keyboard	Input via ASCII keyboard
Switching the operating mode while a block is being edited	Not permitted	Permitted
PGM CALL, **SEL TABLE**, **SEL PATTERN**, **SEL CONTOUR**: Selection of file in a pop-up window	Available	Not available
File handling:		
■ **Save file** function	■ Available	■ Not available
■ **Save file as** function	■ Available	■ Not available
■ Discard changes	■ Available	■ Not available
File management:		
■ Mouse operation	■ Available	■ Available
■ Sorting function	■ Available	■ Available
■ Entry of name	■ Opens the **Select file** pop-up window	■ Synchronizes the cursor
■ Support of short cuts	■ Not available	■ Available
■ Favorites management	■ Not available	■ Available
■ Configuration of column structure	■ Not available	■ Available
■ Soft-key arrangement	■ Slightly different	■ Slightly different
Skip block function	Insert/Remove by soft key	Insert/Remove with ASCII keyboard
Selecting a tool from the table	Selection via split-screen menu	Selection in a pop-up window
Using the cursor in tables	After editing a value, the horizontal arrow keys can be used for positioning within the column	After editing a value, the horizontal arrow keys can be used for positioning to the next/previous column
Programming special functions with the SPEC FCT key	Pressing the key opens a soft-key row as a submenu. To exit the submenu, press the SPEC FCT key again; then the TNC shows the last active soft-key row	Pressing the key adds the soft-key row as the last row. To exit the menu, press the SPEC FCT key again; then the TNC shows the last active soft-key row
Programming approach and departure motions with the APPR DEP key	Pressing the key opens a soft-key row as a submenu. To exit the submenu, press the APPR DEP key again; then the TNC shows the last active soft-key row	Pressing the key adds the soft-key row as the last row. To exit the menu, press the APPR DEP key again; then the TNC shows the last active soft-key row
Pressing the END hard key while the **CYCLE DEF** and **TOUCH PROBE** menus are active	Terminates the editing process and calls the file manager	Exits the respective menu

Function	TNC 320	iTNC 530
Calling the file manager while the **CYCLE DEF** and **TOUCH PROBE** menus are active	Terminates the editing process and calls the file manager. The respective soft-key row remains selected when the file manager is exited	Error message **Key non-functional**
Calling the file manager while **CYCL CALL**, **SPEC FCT**, **PGM CALL** and **APPR/DEP** menus are active	Terminates the editing process and calls the file manager. The respective soft-key row remains selected when the file manager is exited	Terminates the editing process and calls the file manager. The basic soft-key row is selected when the file manager is exited
Datum table:		
■ Sorting function by values within an axis	■ Available	■ Not available
■ Resetting the table	■ Available	■ Not available
■ Hiding axes that are not present	■ Not available	■ Available
■ Switching the list/form view	■ Switchover via split-screen key	■ Switchover by toggle soft key
■ Inserting individual line	■ Allowed everywhere, renumbering possible after request. Empty line is inserted, must be filled with zeros manually	■ Only allowed at end of table. Line with value 0 in all columns is inserted
■ Transfer of actual position values in individual axis to the datum table per keystroke	■ Not available	■ Available
■ Transfer of actual position values in all active axes to the datum table per keystroke	■ Not available	■ Available
■ Using a key to capture the last positions measured by TS	■ Not available	■ Available
■ Entry of comment in **DOC** column	■ With the "Edit the current field" function and the on-line keyboard	■ Via ASCII keyboard
FK free contour programming:		
■ Programming of parallel axes	■ With X/Y coordinates, independent of machine type; switchover with **FUNCTION PARAXMODE**	■ Machine-dependent with the existing parallel axes
■ Automatic correction of relative references	■ Relative references in contour subprograms are not corrected automatically	■ All relative references are corrected automatically

Function	TNC 320	iTNC 530
Handling of error messages:		
■ Help for error messages	■ Call via ERR key	■ Call via HELP key
■ Help for error messages while a block is being edited	■ Cause and corrective action cannot be displayed while highlight is on the block	■ Pop-up window shows cause and corrective action
■ Switching the operating mode while help menu is active	■ Help menu is closed when the operating mode is switched	■ Operating mode switchover is not allowed (key is non-functional)
■ Selecting the background operating mode while help menu is active	■ Help menu is closed when F12 is used for switching	■ Help menu remains open when F12 is used for switching
■ Identical error messages	■ Are collected in a list	■ Are displayed only once
■ Acknowledgment of error messages	■ Every error message (even if it is displayed more than once) must be acknowledged, the **`Delete all`** function is available	■ Error message to be acknowledged only once
■ Access to protocol functions	■ Log and powerful filter functions (errors, keystrokes) are available	■ Complete log without filter functions available
■ Saving service files	■ Available. No service file is created when the system crashes	■ Available. A service file is automatically created when the system crashes
Find function:		
■ List of words recently searched for	■ Not available	■ Available
■ Show elements of active block	■ Not available	■ Available
■ Show list of all available NC blocks	■ Not available	■ Available
Starting the find function with the up/down arrow keys when highlight is on a block	Works with max. 9999 blocks, can be set via config datum	No limitation regarding program length
Programming graphics:		
■ Depiction of the traverse paths of an individual NC block after the graphic was deleted by soft key	■ Not possible; after pressing CLEAR GRAPHIC soft key, all previously defined NC blocks are displayed	■ Available
■ True-to-scale display of grid	■ Available	■ Not available
■ Editing contour subprograms in SLII cycles with AUTO DRAW ON	■ If error messages occur, the cursor is on the **`CYCL CALL`** block in the main program	■ If error messages occur, the cursor is on the error-causing block in the contour subprogram
■ Moving the zoom window	■ Repeat function not available	■ Repeat function available
Programming minor axes:		
■ Syntax **`FUNCTION PARAXCOMP`**: Define the behavior of the display and the paths of traverse	■ Available	■ Not available
■ Syntax **`FUNCTION PARAXMODE`**: Define the assignment of the parallel axes to be traversed	■ Available	■ Not available

Function	TNC 320	iTNC 530
Programming OEM cycles		
■ Access to table data	■ Via **SQL** commands	■ Via **FN17**/**FN18** or **TABREAD**-**TABWRITE** functions
■ Access to machine parameters	■ With the **CFGREAD** function	■ Via **FN18** functions
■ Creating interactive cycles with **CYCLE QUERY**, e.g. touch-probe cycles in Manual Operation mode	■ Available	■ Not available

Comparison: Differences in Test Run, functionality

Function	TNC 320	iTNC 530
Display of delta values **DR** and **DL** from **TOOL CALL** block	Are not considered	Are considered
Test Run up to block N	Function not available	Available function
Calculation of machining time	Each time the simulation is repeated by pressing the START soft key, the machining time is totaled.	Each time the simulation is repeated by pressing the START soft key, time calculation starts at 0.

Comparison: Differences in Test Run, operation

Function	TNC 320	iTNC 530
Arrangement of soft-key rows and soft keys within the rows	Arrangement of soft-key rows and soft-keys varies depending on the active screen layout.	
Zoom function	Each sectional plane can be selected by individual soft keys	Sectional plane can be selected via three toggle soft keys
Character set for PROGRAM screen layout	Small character set	Medium character set
Performing a Test Run in Single block mode, switching to the Programming mode of operation at any time	When you switch to the Programming mode of operation, the warning **No write permission** is displayed; once a change has been made, the error message is cleared and the program is reset to the beginning when you switch back to Test Run mode.	The operating mode can be switched. Changes to the program do not influence the position of the cursor.
Machine-specific miscellaneous functions M	Lead to error messages if they are not integrated in the PLC	Are ignored during Test Run
Displaying/editing the tool table	Function available via soft key	Function not available

Comparison: Differences in Manual Operation, functionality

Function	TNC 320	iTNC 530
3-D ROT function: Manual deactivation of the Tilt working plane function	If the tilted working plane function is deactivated for both operating modes, the text fields will be filled with zeros instead of the current rotary axis positions when the 3-D ROT function is next called. The positions are entered correctly if only one operating mode is set to **inactive**.	The programmed values are displayed in the 3-D ROT dialog even if the Tilt working plane function is set to **inactive** for both operating modes.
Jog increment function	The jog increment can be defined separately for linear and rotary axes.	The jog increment applies for both linear and rotary axes.
Preset table	Basic transformation (translation and rotation) of machine table system to workpiece system via the columns **X**, **Y** and **Z**, as well as spatial angles **SPA**, **SPB** and **SPC**. In addition, the columns **X_OFFS** to **W_OFFS** can be used to define the axis offset of each individual axis. The function of the axis offsets can be configured.	Basic transformation (translation) of machine table system to workpiece system via the columns **X**, **Y** and **Z**, as well as a **ROT** basic rotation in the working plane (rotation). In addition, the columns **A** to **W** can be used to define datums in the rotary and parallel axes.
Behavior during presetting	Presetting in a rotary axis has the same effect as an axis offset. The offset is also effective for kinematics calculations and for tilting the working plane. The machine parameter **CfgAxisPropKin->presetToAlignAxis** is used to define whether the axis offset is to be taken into account internally after zero setting. Independently of this, an axis offset has always the following effects: ■ An axis offset always influences the nominal position display of the affected axis (the axis offset is subtracted from the current axis value). ■ If a rotary axis coordinate is programmed in an L block, then the axis offset is added to the programmed coordinate.	Rotary axis offsets defined by machine parameters do not influence the axis positions that were defined in the Tilt working plane function. MP7500 bit 3 defines whether the current rotary axis position referenced to the machine datum is taken into account, or whether a position of 0° is assumed for the first rotary axis (usually the C axis).

Function	TNC 320	iTNC 530
Handling of preset table:		
■ Editing the preset table in the Programming mode of operation	■ Possible	■ Not possible
■ Preset tables that depend on the range of traverse	■ Not available	■ Available
■ Entry of comment in DOC column	■ Via online keyboard	■ Via ASCII keyboard
Definition of feed-rate limitation	Feed-rate limitation can be defined separately for linear and rotary axes	Only one feed-rate limitation can be defined for linear and rotary axes

Comparison: Differences in Manual Operation, operation

Function	TNC 320	iTNC 530
Character set for POSITION screen layout	Small position display	Large position display
Capturing the position values from mechanical probes	Actual-position capture by soft key	Actual-position capture by hard key
Exiting the touch probe functions menu	Only via the END soft key	Via the END soft key or the END hard key
Exit the preset table	Only through the BACK/END soft keys	Through the END hard key at any time
Multiple editing of tool table TOOL.T, or pocket table tool_p.tch	Soft-key row that was last active before exiting is active	Permanently defined soft-key row (soft-key row 1) is displayed

Comparison: Differences in Program Run, operation

Function	TNC 320	iTNC 530
Arrangement of soft-key rows and soft keys within the rows	Arrangement of soft-key rows and soft-keys varies depending on the active screen layout	
Character set for PROGRAM screen layout	Small character set	Medium character set
Editing of program after program run was interrupted by switching to the Single block mode of operation	The INTERNAL STOP soft key must also be pressed to cancel the program	Editing is possible right after switching to the **`Programming`** mode of operation
Operating-mode switchover after program run was interrupted by switching to the Single block mode of operation	The INTERNAL STOP soft key must also be pressed to cancel the program	Switching the operating mode is allowed
Operating-mode switchover after program run was interrupted by switching to the Single block mode of operation, and canceled by **`INTERNAL STOP`** on the TNC 320	When you return to the Program Run mode of operation: Error message **`Selected block not addressed`**. Use mid-program startup to select the point of interruption	Switching the operating mode is allowed, modal information is saved, program run can be continued by pressing NC start
GOTO is used to go to FK sequences after program run was interrupted there before switching the operating mode	Error message **`FK programming: Undefined starting position`**	GOTO allowed
Mid-program startup:		
■ Behavior after restoring the machine status	■ The menu for returning must be selected with the RESTORE POSITION soft key	■ Menu for returning is selected automatically
■ Returning to the point of interruption with positioning logic	■ The order of axis approach cannot be recognized; a fixed sequence of axes is always displayed on the screen	■ The order of axis approach is displayed on the screen by highlighting the corresponding axes
■ Completing positioning for mid-program startup	■ After position has been reached, positioning mode must be exited with the RESTORE POSITION soft key	■ The positioning mode is automatically exited after the position has been reached
■ Switching the screen layout for mid-program startup	■ Only possible, if startup position has already been approached	■ Possible in all operating states

Function	TNC 320	iTNC 530
Error messages	Error messages (e.g. limit switch messages) are still active after the error has been corrected and must be acknowledged separately	Error messages are sometimes acknowledged automatically after the error has been corrected
Editing Q-parameter contents after program run was interrupted by switching to the Single block mode of operation	The INTERNAL STOP soft key must also be pressed to cancel the program	Direct editing possible
Manual traverse during program interruption and with active **M118**	Function not available	Available function

Comparison: Differences in Program Run, traverse movements

Caution: Check the traverse movements!

NC programs that were created on earlier TNC controls may lead to different traverse movements or error messages on a TNC 320!

Be sure to take the necessary care and caution when running-in programs!

Please find a list of known differences below. The list does not pretend to be complete!

Function	TNC 320	iTNC 530
Handwheel-superimposed traverse with M118	Effective in the active coordinate system (which may also be rotated or tilted), or in the machine-based coordinate system, depending on the setting in the 3-D ROT menu for manual operation	Effective in the machine-based coordinate system
M118 in conjunction with M128	Function not available	Available function
Approach/Departure with **APPR**/**DEP**, **R0** is active, contour element plane is not equal to working plane	If possible, the blocks are executed in the defined **contour element plane**, error message for **APPRLN**, **DEPLN**, **APPRCT**, **DEPCT**	If possible, the blocks are executed in the defined **working plane**; error message for **APPRLN**, **APPRLT**, **APPRCT**, **APPRLCT**
Scaling approach/departure movements (**APPR**/**DEP**/**RND**)	Axis-specific scaling factor is allowed, radius is not scaled	Error message
Approach/departure with **APPR**/**DEP**	Error message if **R0** is programmed for **APPR**/**DEP LN** or **APPR**/**DEP CT**	Tool radius 0 and compensation direction **RR** are assumed
Approach/departure with **APPR**/**DEP** if contour elements with length 0 are defined	Contour elements with length 0 are ignored. The approach/departure movements are calculated for the first or last valid contour element	An error message is issued if a contour element with length 0 is programmed after the **APPR** block (relative to the first contour point programmed in the APPR block) For a contour element with length 0 before a **DEP** block, the TNC does not issue an error message, but uses the last valid contour element to calculate the departure movement
Effect of Q parameters	**Q60** to **Q99** (or **QS60** to **QS99**) are always local.	**Q60** to **Q99** (or **QS60** to **QS99**) are local or global, depending on MP7251 in converted cycle programs (.cyc). Nested calls may cause problems

Function	TNC 320	iTNC 530
Automatic cancelation of tool radius compensation	■ Block with **R0** ■ **DEP** block ■ **END PGM**	■ Block with **R0** ■ **DEP** block ■ **PGM CALL** ■ Programming of Cycle 10 **ROTATION** ■ Program selection
NC blocks with **M91**	No consideration of tool radius compensation	Consideration of tool radius compensation
Tool shape compensation	Tool shape compensation is not supported, because this type of programming is considered to be axis-value programming, and the basic assumption is that axes do not form a Cartesian coordinate system	Tool shape compensation is supported
Paraxial positioning blocks	Radius compensation as in **L** blocks	The tool approaches from the current position of the previous block to the programmed coordinate value. If the next block is a linear block, it is dealt with in the same way as an additional radius-compensation block so that the path will be contour-parallel from the next but one linear block
Mid-program startup in a point table	The tool is positioned above the next position to be machined	The tool is positioned above the last position that has been completely machined
Empty **CC** block (pole of last tool position is used) in NC program	Last positioning block in the working plane must contain both coordinates of the working plane	Last positioning block in the working plane does not necessarily need to contain both coordinates of the working plane. Can cause problems with **RND** or **CHF** blocks
Axis-specific scaling of **RND** block	**RND** block is scaled, the result is an ellipse	Error message is issued
Reaction if a contour element with length 0 is defined before or after a **RND** or **CHF** block	Error message is issued	Error message is issued if a contour element with length 0 is located before the **RND** or **CHF** block Contour element with length 0 is ignored if the contour element with length 0 is located after the **RND** or **CHF** block
Circle programming with polar coordinates	The incremental rotation angle **IPA** and the direction of rotation **DR** must have the same sign. Otherwise, an error message will be issued	The algebraic sign of the direction of rotation is used if the sign defined for **DR** differs from the one defined for **IPA**

Function	TNC 320	iTNC 530
Tool radius compensation on circular arc or helix with angular length = 0	The transition between the adjacent elements of the arc/helix is generated. Also, the tool axis motion is executed right before this transition. If the element is the first or last element to be corrected, the next or previous element is dealt with in the same way as the first or last element to be corrected	The equidistant line of the arc/helix is used for generating the tool path
Checking the algebraic sign of the depth parameter in fixed cycles	Must be deactivated if Cycle 209 is used	No restrictions
Tool change while tool radius compensation is active	Program cancelation with error message	Tool radius compensation is canceled, tool change is performed
Compensation of tool length in the position display	The values **L** and **DL** from the tool table and the value **DL** from the **TOOL CALL** are taken into account in the position display	The values **L** and **DL** from the tool table are taken into account in the position display

Function	TNC 320	iTNC 530
SLII Cycles 20 to 24:		
■ Number of definable contour elements	■ Max. 16384 blocks in up to 12 subcontours	■ Max. 8192 contour elements in up to 12 subcontours, no restrictions for subcontour
■ Define the working plane	■ Tool axis in **TOOL CALL** block defines the working plane	■ The axes of the first positioning block in the first subcontour define the working plane
■ Traverse paths during rough-out	■ Islands are not circumnavigated. Reciprocating plunge infeed at reduced feed rate (increase in machining time)	■ Islands are circumnavigated at the current machining depth
■ Contour-parallel rough-out, or paraxial channel milling and rough-out	■ Rough-out is always contour-parallel	■ Configurable via MP7420
■ Internal consideration of combined contours	■ Combinations always refer to the defined uncompensated contour	■ With MP7420, you can define whether the uncompensated or compensated contour is to be combined
■ Rough-out strategy if multiple pockets are defined	■ At first, all pockets are roughed-out on the same plane	■ With MP7420, you can define whether individual pockets are roughed-out completely or on the same plane
■ Position at end of SL cycle	■ End position = clearance height above the last position that is defined before the cycle call	■ With MP7420, you can define whether the end position is above the last programmed position, or whether the tool moves only to clearance height
■ Tangential arcs for floor finishing Cycle 23	■ Curvature of tangential arcs is derived from the curvature of the target contour. To position the circular arc, the target contour is systematically searched from end to beginning until a position is found where no collision can occur. If this is not possible, the arc length is halved until it can be positioned	■ Circular arcs are generated between the starting point of the outermost path of the roughing tool and the center of the first contour element of the path of the finishing tool
■ Tangential arcs for side finishing Cycle 24	■ The max. width of the arc is three tool radii, the max. angular length is 0.8 rad. To position the circular arc, the target contour is systematically searched from end to beginning until a position is found where no collision can occur. If this is not possible, the arc length is halved until it can be positioned	■ Max. width of the arc (tool moves backward on tangential arc from starting point of the path to shortly before next edge contour), max. arc height: finishing allowance + safety clearance

Function	TNC 320	iTNC 530
SLII Cycles 20 to 24:		
■ Handling of coordinates and axis values outside the working plane	■ Error message is issued	■ Axes that are outside the working plane in the contour description are ignored
■ Handling of islands which are not contained in pockets	■ Cannot be defined with complex contour formula	■ Restricted definition in complex contour formula is possible
■ Set operations for SL cycles with complex contour formulas	■ Real set operation possible	■ Only restricted performance of real set operation possible
■ Radius compensation is active during **CYCL CALL**	■ Error message is issued	■ Radius compensation is canceled, program is executed
■ Paraxial positioning blocks in contour subprogram	■ Error message is issued	■ Program is executed
■ Miscellaneous functions **M** in contour subprogram	■ Error message is issued	■ M functions are ignored
■ Infeed movements in contour subprogram	■ Error message is issued	■ Infeed movements are ignored
■ **M110** (feed-rate reduction for inside corner)	■ Function does not work within SL cycles	■ Function also works within SL cycles
SLII Contour Train Cycle 25: **APPR/DEP** blocks in contour definition	Not allowed, machining of closed contours is more coherent	**APPR/DEP** blocks are allowed as contour elements
General **cylinder surface machining**:		
■ Contour definition	■ With X/Y coordinates, independent of machine type	■ Machine-dependent, with existing rotary axes
■ Offset definition on cylinder surface	■ With datum shift in X/Y, independent of machine type	■ Machine-dependent datum shift in rotary axes
■ Offset definition for basic rotation	■ Available function	■ Function not available
■ Circle programming with C/CC	■ Available function	■ Function not available
■ **APPR/DEP** blocks in contour definition	■ Function not available	■ Available function
Cylinder surface machining with Cycle 28:		
■ Complete roughing-out of slot	■ Available function	■ Function not available
■ Definable tolerance	■ Available function	■ Available function
Cylinder surface machining with Cycle 29	Direct plunging to contour of ridge	Circular approach to contour of ridge
Cycles 25x for pockets, studs and slots	In limit ranges (geometrical conditions of tool/contour) error messages are triggered if plunging movements lead to unreasonable/critical behavior	In limit ranges (geometrical conditions of tool/contour), vertical plunging is used if required

Function	TNC 320	iTNC 530
Touch probe cycles for datum setting (manual and automatic cycles)	Cycles can only be executed if the tilted working plane function is inactive, the datum shift is inactive and rotation with Cycle 10 is inactive. As of version 34055x 05, touch probe cycles with active coordinate transformations can be used.	No restrictions in connection with coordinate transformations
PLANE function:		
■ **TABLE ROT**/**COORD ROT** not defined	■ Configured setting is used	■ **COORD ROT** is used
■ Machine is configured for axis angle	■ All **PLANE** functions can be used	■ Only **PLANE AXIAL** is executed
■ Programming an incremental spatial angle according to **PLANE AXIAL**	■ Error message is issued	■ Incremental spatial angle is interpreted as an absolute value
■ Programming an incremental axis angle according to **PLANE SPATIAL** if the machine is configured for spatial angle	■ Error message is issued	■ Incremental axis angle is interpreted as an absolute value
Special functions for cycle programming:		
■ FN17	■ Function available, details are different	■ Function available, details are different
■ FN18	■ Function available, details are different	■ Function available, details are different

Comparison: Differences in MDI operation

Function	TNC 320	iTNC 530
Execution of connected sequences	Function partially available	Available function
Saving modally effective functions	Function partially available	Available function

Comparison: Differences in programming station

Function	TNC 320	iTNC 530
Demo version	Programs with more than 100 NC blocks cannot be selected, an error message is issued	Programs can be selected, max. 100 NC blocks are displayed, further blocks are truncated in the display
Demo version	If nesting with PGM CALL results in more than 100 NC blocks, there is no test graphic display; an error message is not issued	Nested programs can be simulated.
Copying NC programs	Copying to and from the directory **TNC:** is possible with Windows Explorer	TNCremo or file manager of programming station must be used for copying
Shifting the horizontal soft-key row	Clicking the soft-key bar shifts the soft-key row to the right, or to the left	Clicking any soft-key bar activates the respective soft-key row

DIN/ISO Function Overview for TNC 320

M functions	
M00	Program STOP/Spindle STOP/Coolant OFF
M01	Optional program STOP
M02	STOP program run/Spindle STOP/Coolant OFF/CLEAR status display (depending on machine parameter)/Go to block 1
M03	Spindle ON clockwise
M04	Spindle ON counterclockwise
M05	Spindle STOP
M06	Tool change/STOP program run (depending on machine parameter)/Spindle STOP
M08	Coolant ON
M09	Coolant OFF
M13	Spindle ON clockwise/Coolant ON
M14	Spindle ON counterclockwise/Coolant ON
M30	Same function as M02
M89	Vacant miscellaneous function or Cycle call, modally effective (depending on machine parameter)
M99	Blockwise cycle call
M91	Within the positioning block: Coordinates are referenced to machine datum
M92	Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position
M94	Reduce the rotary axis display to a value below 360°
M97	Machine small contour steps
M98	Machine open contours completely
M109	Constant contouring speed at tool cutting edge (increase and decrease feed rate)
M110	Constant contouring speed at tool cutting edge (feed rate decrease only)
M111	Reset M109/M110
M116	Feed rate for rotary axes in mm/min
M117	Reset M116
M118	Superimpose handwheel positioning during program run
M120	Pre-calculate radius-compensated contour (LOOK AHEAD)
M126	Shortest-path traverse of rotary axes
M127	Reset M126
M128	Retain position of tool tip when positioning tilting axes (TCPM)
M129	Reset M128

M functions	
M130	Within the positioning block: Points are referenced to the untilted coordinate system
M140	Retraction from the contour in the tool-axis direction
M141	Suppress touch probe monitoring
M143	Delete basic rotation
M148	Retract the tool automatically from the contour at NC stop
M149	Reset M148

G Functions

Tool movements

G00	Straight-line interpolation, Cartesian coordinates, rapid traverse
G01	Straight-line interpolation, Cartesian coordinates
G02	Circular interpolation, Cartesian coordinates, clockwise
G03	Circular interpolation, Cartesian coordinates, counterclockwise
G05	Circular interpolation, Cartesian coordinates, without indication of direction
G06	Circular interpolation, Cartesian coordinates, tangential contour approach
G07*	Paraxial positioning block
G10	Straight-line interpolation, polar coordinates, rapid traverse
G11	Straight-line interpolation, polar coordinates
G12	Circular interpolation, polar coordinates, clockwise
G13	Circular interpolation, polar coordinates, counterclockwise
G15	Circular interpolation, polar coordinates, without indication of direction
G16	Circular interpolation, polar coordinates, tangential contour approach

Chamfer/Rounding/Approach contour/Depart contour

G24*	Chamfer with length R
G25*	Corner rounding with radius R
G26*	Tangential contour approach with radius R
G27*	Tangential contour approach with radius R

Tool definition

G99*	With tool number T, length L, radius R

Tool radius compensation

G40	No tool radius compensation
G41	Tool radius compensation, left of the contour
G42	Tool radius compensation, right of the contour
G43	Paraxial compensation for G07, lengthening
G44	Paraxial compensation for G07, shortening

G Functions

Blank form definition for graphics

G30	(G17/G18/G19) min. point
G31	(G90/G91) max. point

Cycles for drilling, tapping and thread milling

G240	Centering
G200	Drilling
G201	Reaming
G202	Boring
G203	Universal drilling
G204	Back boring
G205	Universal pecking
G206	Tapping with a floating tap holder
G207	Rigid tapping
G208	Bore milling
G209	Tapping with chip breaking
G241	Single-lip deep-hole drilling

Cycles for drilling, tapping and thread milling

G262	Thread milling
G263	Thread milling/countersinking
G264	Thread drilling/milling
G265	Helical thread drilling/milling
G267	External thread milling

Cycles for milling pockets, studs and slots

G251	Rectangular pocket, complete
G252	Circular pocket, complete
G253	Slot, complete
G254	Circular slot, complete
G256	Rectangular stud
G257	Circular stud

Cycles for creating point patterns

G220	Circular point pattern
G221	Point patterns on lines

SL cycles, group 2

G37	Contour geometry, list of subcontour program numbers
G120	Contour data (applies to G121 to G124)
G121	Pilot drilling
G122	Rough-out
G123	Floor finishing
G124	Side finishing
G125	Contour train (machining open contour)
G127	Cylinder surface
G128	Cylindrical surface slot

Coordinate transformation

G53	Datum shift in datum table
G54	Datum shift in program
G28	Mirror image
G73	Rotation of the coordinate system
G72	Scaling factor (reduce or enlarge contour)
G80	Tilting the working plane
G247	Datum setting

G Functions

Cycles for multipass milling

G230	Multipass milling of smooth surfaces
G231	Multipass milling of tilted surfaces
G232	Face milling

*) Non-modal function

Touch probe cycles for measuring workpiece misalignment

G400	Basic rotation using two points
G401	Basic rotation from two holes
G402	Basic rotation from two studs
G403	Compensate a basic rotation via a rotary axis
G404	Set basic rotation
G405	Compensating misalignment with the C axis

Touch probe cycles for datum setting

G408	Slot center reference point
G409	Reference point at center of hole
G410	Datum from inside of rectangle
G411	Datum from outside of rectangle
G412	Datum from inside of circle
G413	Datum from outside of circle
G414	Datum in outside corner
G415	Datum in inside corner
G416	Datum circle center
G417	Datum in touch probe axis
G418	Datum in center of 4 holes
G419	Reference point in selectable axis

Touch probe cycles for workpiece measurement

G55	Measure any coordinate
G420	Measure any angle
G421	Measure hole
G422	Measure cylindrical stud
G423	Measure rectangular pocket
G424	Measure rectangular stud
G425	Measure slot
G426	Measure ridge
G427	Measure any coordinate
G430	Measure circle center
G431	Measure any plane

Touch probe cycles for tool measurement

G480	Calibrate the TT
G481	Measure tool length
G482	Measure tool radius
G483	Measure tool length and tool radius

Special cycles

G04*	Dwell time with F seconds
G36	Spindle orientation
G39*	Program call
G62	Tolerance deviation for fast contour milling
G440	Measure axis shift
G441	Fast probing

G Functions

Define machining plane

G17	Working plane X/Y, tool axis Z
G18	Working plane Z/X, tool axis Y
G19	Working plane Y/Z, tool axis X
G20	Tool axis IV

Dimensions

G90	Absolute dimensions
G91	Incremental dimensions

Unit of measure

G70	Inches (set at start of program)
G71	Millimeters (set at start of program)

Other G functions

G29	Transfer the last nominal position value as a pole (circle center)
G38	STOP program run
G51*	Next tool number (with central tool file)
G79*	Cycle call
G98*	Set label number

*) Non-modal function

Addresses

%	Program beginning
%	Program call
#	Datum number with G53
A	Rotation about X axis
B	Rotation about Y axis
C	Rotation about Z axis
D	Q-parameter definitions
DL	Length wear compensation with T
DR	Radius wear compensation with T
E	Tolerance with M112 and M124
F	Feed rate
F	Dwell time with G04
F	Scaling factor with G72
F	Factor for feed-rate reduction F with M103
G	G Functions
H	Polar coordinate angle
H	Rotation angle with G73
H	Tolerance angle with M112
I	X coordinate of the circle center/pole
J	Y coordinate of the circle center/pole
K	Z coordinate of the circle center/pole

Addresses

L	Set a label number with G98
L	Jump to a label number
L	Tool length with G99
M	M functions
N	Block number
P	Cycle parameters in machining cycles
P	Value or Q parameter in Q-parameter definition
Q	Q parameter
R	Polar coordinate radius
R	Circular radius with G02/G03/G05
R	Rounding radius with G25/G26/G27
R	Tool radius with G99
S	Spindle speed
S	Oriented spindle stop with G36
T	Tool definition with G99
T	Tool call
T	Next tool with G51
U	Axis parallel to X axis
V	Axis parallel to Y axis
W	Axis parallel to Z axis
X	X axis
Y	Y axis
Z	Z axis
*	End of block

Contour cycles

Sequence of Program Steps for Machining with Several Tools	
List of subcontour programs	G37 P01 ...
Define contour data	G120 Q1 ...
Define/Call **drill** Contour cycle: pilot drilling Cycle call	G121 Q10 ...
Define/Call **roughing mill** Contour cycle: rough-out Cycle call	G122 Q10 ...
Define/Call **finishing mill** Contour cycle: floor finishing Cycle call	G123 Q11 ...
Define/Call **finishing mill** Contour cycle: side finishing Cycle call	G124 Q11 ...
End of main program, return	**M02**
Contour subprograms	G98 ... G98 L0

Radius compensation of the contour subprograms

Contour	Programming Sequence of the Contour Elements	Radius Compen-sation
Internal (pocket)	Clockwise (CW) Counterclockwise (CCW)	G42 (RR) G41 (RL)
External (island)	Clockwise (CW) Counterclockwise (CCW)	G41 (RL) G42 (RR)

Coordinate transformation

Coordinate transformation	Activate	Cancelation
Datum shift	G54 X+20 Y+30 Z+10	G54 X0 Y0 Z0
Mirror image	G28 X	G28
Rotation	G73 H+45	G73 H+0
Scaling factor	G72 F 0.8	G72 F1
Working plane	G80 A+10 B+10 C+15	G80
Working plane	PLANE ...	PLANE RESET

Q-parameter definitions

D	Function
00	Assignment
01	Addition
02	Subtraction
03	Multiplication
04	Division
05	Root
06	Sine
07	Cosine
08	Root sum of squares $c = \sqrt{a^2+b^2}$
09	If equal, go to label number
10	If not equal, go to label number
11	If greater than, go to label number
12	If less than, go to label number
13	Angle from c sin a and c cos a
14	Error number
15	Print
19	Assignment PLC

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 5061
E-mail: info@heidenhain.de

Technical support FAX +49 8669 32-1000
Measuring systems ☏ +49 8669 31-3104
E-mail: service.ms-support@heidenhain.de
TNC support ☏ +49 8669 31-3101
E-mail: service.nc-support@heidenhain.de
NC programming ☏ +49 8669 31-3103
E-mail: service.nc-pgm@heidenhain.de
PLC programming ☏ +49 8669 31-3102
E-mail: service.plc@heidenhain.de
Lathe controls ☏ +49 8669 31-3105
E-mail: service.lathe-support@heidenhain.de

www.heidenhain.de

679 226-21 · Ver01 · SW05 · Printed in Germany · 11/2011 · F&W